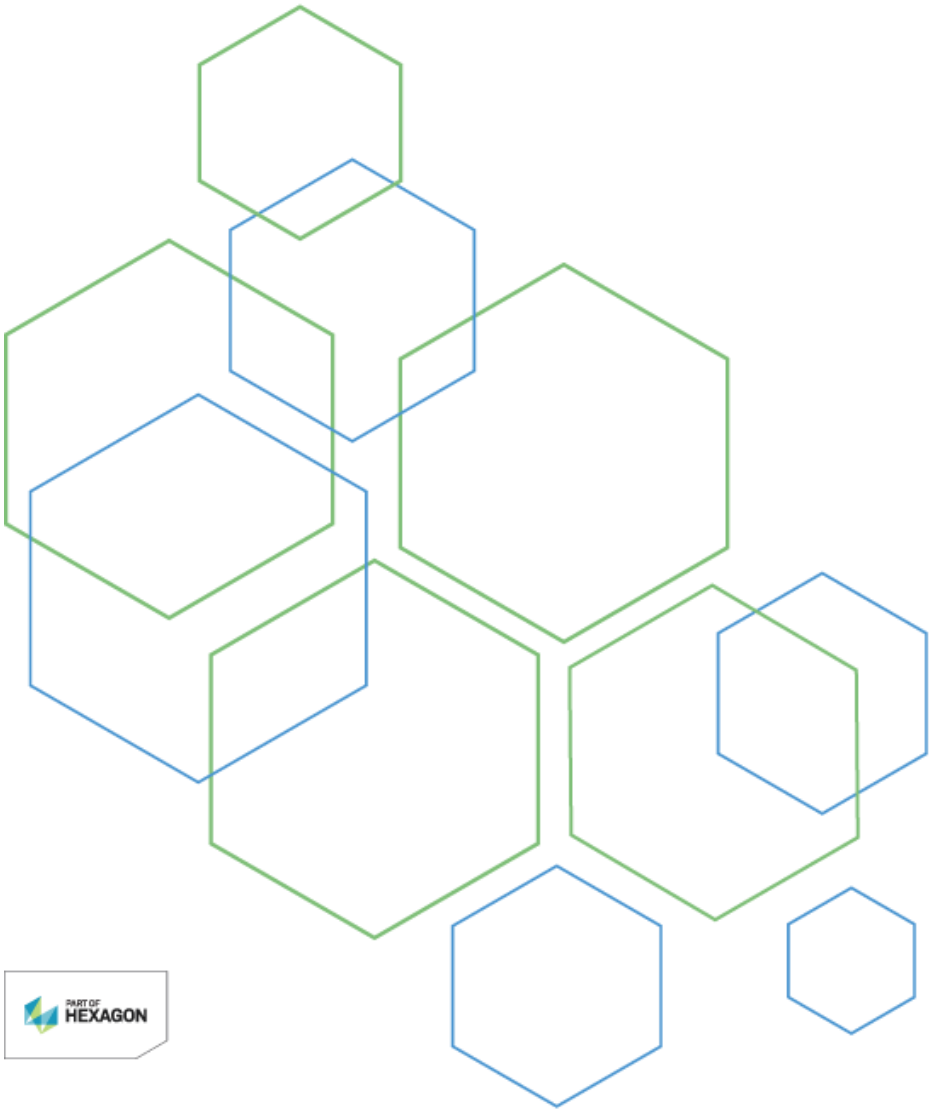
INTERGRAPH Smart 3D

Molded Forms

User's Guide



PART OF HEXAGON

Version 2016 (11.0)
November 2016

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Preface

This document is a user's guide for the Molded Forms functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (*https://smartsupport.intergraph.com*).

What's New in Molded Forms

The following changes have been made to the Molded Forms task.

Version 2016 (11.0)

- Formerly known as SmartPlant 3D (SP3D) and SmartMarine 3D (SM3D), the products are now integrated and rebranded as Smart 3D (S3D). Replaced all instances of SP3D and SM3D with S3D in the **Export Neutral/GeniE XML** section of this guide. (P1 CP:273062)
- The **Bracket Plate System** command improves the workflow for bracket creation. **Length 1** and **Length 2** boxes do not display by default when **Support 1** and **Support 2** are planar. For curved supports, **Length 1** and **Length 2** boxes control the bracket plane position, but not bracket creation. Bracket browser and rule creation icons are improved. For more information, see *Bracket Plate System Ribbon* in *Bracket Plate System* (on page 227). (P1 CP:271139, P1 CP:244914, P3 CP:251341)
- When you copy a plate system using **Delete Optional** in the **Paste** command, the software replaces the plate with a standalone boundary to prevent the plate from going to the **To Do List**. For more information, see *Placing Plate Systems* (on page 96). (P1 CP:210300)
- **Delete Optional** in the **Paste** command supports plates, openings, profile systems, tripping stiffeners, seams, profile seams, and reference curves. (P1 CP:210300, CP:210328, CP:210340, CP:210345, CP:210378, CP:210388, CP:210397, CP:261307, CP:262625)
- **Model Data Transform** supports plates, profile systems, tripping stiffeners, profile seams, topological openings, and sketched openings. (P1 CP:231434, CP:231435, CP:231439, CP:231448, CP:258168)
- You can create multiple plate systems with appropriate macros in the **Advanced Plate System** command. (P1 CP:245070, P2 CP:259821)
- Export to Neutral Schema now supports mapping of custom attributes on the Smart 3D objects, and exporting these attributes to the neutral schema. New class map sheets are added to map these attributes. For more information, see *Export Neutral/GeniE XML (File Menu)* (on page 1241) and *ExportAttributeMapping.xls* (on page 1246). (P2 CP:218559)
- A new option, **Non Planar**, has been added to **Plate Systems** under **External Geometry Export Options** in the **Export NSchema/GeniE** dialog box. For more information, see *Export NSchema/GeniE Dialog Box* (on page 1244). (P2 CP:274229)

- Information related to exporting of **Non Planar** plate systems has been added. For more information, see *Export data to Neutral XML* (on page 1242). (P2 CP:274229)
- **Welded**, **Snip**, **Cutback**, and **Bracketed** are now available as options for defining the profile connection types. For more information, see *PDBT MF Section Orientation Tab* (see "*Section Orientation Tab (Profile Properties Dialog Box)*" on page 732). (P2 CP:264507, CP:280640)
- Updated *Objects not supported by Copy by Family* (on page 1302), *Objects and behaviors supported in Attribute Modify* (on page 1303), and *Solutions to common problems in Copy by Family* (on page 1310). (P2 CP:237601, CP:211489, CP:211490, CP:213977, CP:238915, CP:238917, CP:253294)
- The naming rule is improved for a plate system that is not on a reference plane. For more information, see *Main Tab (Planar Plate System Properties Dialog Box)* (on page 107). (P2 CP:256932)
- A new property, **CoG Coordinate System**, is available for defining center of gravity properties. For more information, see *Weight & CG Tab* (on page 129). (P2 CR:236999)
- Plane controls are simplified. The controls are used in multiple commands for defining planes. For more information, see *Plane Methods* (on page 76). (P2: 244830)
- The **Boundary List** dialog box supports offset values, enhanced display of boundaries in the graphic view, display of standalone planes, and improved boundary list controls. For more information, see *Boundary List Dialog Box* (on page 83). (P2 CP:230926, CP:262705, CP:297356)
- The **Extend Sketch Boundary in Graphic View** option on the **Boundary List** dialog box allows you to extend a sketched boundary without entering Sketch 2D. For more information, see *Boundary List Dialog Box* (on page 83) and *Planar Plate System* (on page 99). (P2 CP:274205)
- Added descriptions of molded conventions for the individual plates of a designed (built-up) member. For more information, see *Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box - Designed Member)* (on page 840). (P2 CP:264474)
- Expanded the descriptions of molded conventions for plate systems. As an example, see *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110). (P2 CP:264474)
- The workflow for offsetting a seam using the **Profile Seam** command is improved. For more information, see *Profile Seam* (on page 914). (P2 CP:264024)
- To improve usability, commonly used properties are available on the plate system command ribbons. For more information, see *Planar Plate System* (on page 99), *Imported Plate System* (on page 130), *Linear Extruded Plate System* (on page 156), *Nonlinear Extruded Plate System* (on page 180), *Revolved Plate System* (on page 205), *Bracket Plate System* (on page 227), *Advanced Plate System* (on page 275), *Child Plate System* (on page 426), and *Ruled Plate System* (on page 449). (P2 CP:276091, CP:292407)
- To improve usability, commonly used properties are available on the profile system command ribbons. For more information, see *Stiffener Profile System by Intersection* (on page 723), *Stiffener Profile System by Offset* (on page 739), *Stiffener Profile System by Projection* (on page 746), *Stiffener Profile System by Table* (on page 755), and *Tripping Stiffener System* (on page 765) (P2 CP:276090,CP:292407)
- The **Subtype** plate system property specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or profile naming. For more information, see *Planar Plate System* (on page 99), *Imported Plate System* (on page 130),

Linear Extruded Plate System (on page 156), *Nonlinear Extruded Plate System* (on page 180), *Revolved Plate System* (on page 205), *Bracket Plate System* (on page 227), *Advanced Plate System* (on page 275), *Child Plate System* (on page 426), and *Ruled Plate System* (on page 449). (P2 CP:227200)

- Added the **Copy MfgData** option to the **Across Model Family Definition** dialog box. For more information, see *Across Model Family Definition Dialog Box* (on page 1281). (P2 CP:245086)
- Added the **Submit Batch Job** command to run batch processes using the Intergraph Batch Services framework. For more information, see *Submit Batch Job* (on page 1380). (P2 CP:273093, P2 CP:178040)
- Added a new locate filter, **Construction Graphics**. For more information, see *Selecting Objects* (on page 40). (P2 CP:271166)
- Updated *Objects and behaviors supported for Copy by Family* (on page 1294), *Objects not supported by Copy by Family* (on page 1302), and *Objects and behaviors supported in Attribute Modify* (on page 1303) for Version 2016. (P3 CP:288254)
- The **Description** option has been removed for bracket plate systems. For more information, see *Main Tab (Bracket System Properties Dialog Box)* (on page 250). (P2 CP:272877)
- The **Automatic Rebound** command is now named **Automatic Reconnect** and reconnects boundaries and definition planes on copied objects. You can also filter the workspace to highlight objects with definition planes and boundaries defined by standalone planes. For more information, see *Automatic Reconnect (Tools Menu)* (on page 1388),*Use filters to view copied objects requiring reconnection* (on page 1388), and *Plane Tab (Automatic Reconnect Dialog Box)* (on page 1394). (P2 CP:271267, CP:259014, CP: 273850)
- A profile can be split by a penetrating standard member (P2 CP:177250)
- Openings migrated from the previous version of the software maintain their sketches and constraints (P2 CP:178057)
- You can select a standard member as a support for a bracket by supports (P2 CP:187594)
- Plate systems can be split by planning seams in addition to design seams. (P2 CP: 229642)
- When creating plate systems, gaps between boundaries display with a glyph to aid in identifying the gap. (P2 CP:244838)
- WBS commands support Molded Forms objects (P2 CP:268662)
- Filters are available on the **Select by Filter** dialog box to select objects by plate thickness, profile cross-section name, plate and profile material, and plate and profile grade. (P2 CP:273852, CP:273853, CP:273854)
- Added information about the options that you can change on root edge reinforcement systems and leaf edge reinforcement systems. For more information, see *Section Orientation Tab (Edge Reinforcement Properties Dialog Box)* (on page 777). (P3 CP:254962)
- Added information about which objects Smart 3D copies as well as what types of modifications are supported by **Copy Symmetry**, **Copy Similar**, and **Copy Across Model**. For more information, see the following:

  Frame mapping and Range Growth Values in Copy by Family (on page 1292)
  Objects and behaviors supported for Copy by Family (on page 1294)
  Objects not supported by Copy by Family (on page 1302)
  Objects and behaviors supported in Attribute Modify (on page 1303)
  Solutions to common problems in Copy by Family (on page 1310)

(P3 CP:263679)

- Expanded **Advanced Plate System** help to include an overview, ribbon description, and workflow for each macro. For more information, see *Advanced Plate Palette Dialog Box* (on page 283). (P3 CP:253349)
- After a command using geometric constructions is interrupted and restarted, you can recover geometric constructions created before the interruption. For more information, see *Geometric Constructions* (on page 1027). (P3 CP:257735)
- **Import** and **Export** commands and error display are available for geometric constructions. For more information, see *Geometric Construction Explorer* (on page 1214). (P3 CP:185171)
- You can add a profile seam to a stiffener profile system by offset (P3 CP:187396)
- Concurrent highlighting of selected boundary objects in the **Workspace Explorer** and graphic view is improved. (P3 CP:251413)
- When a plate system with openings is rebounded in a way that the openings will fall outside of the plate boundaries, the plate modifications complete, and the software places the openings on the **To Do List**. (P3 CP:272150)
- Added information on the **Pick Edges**  option for selecting boundaries. For more information, see *Boundary Methods* (on page 80). (P3 CP:289423)
- For piping and equipment objects in plant mode, **Insulation**, **Equipment Hole**, **Maintenance**, and **Operation** aspects displaying in the 3D graphic view also display in the Sketch 2D environment. In marine mode, the additional aspects also display in Sketch 2D for **Openings**. For more information, see *Format View Dialog Box* in the *Common User's Guide*. (P4 CP: 119768)
- Updated the descriptions of the **Thickening Technique Requested** and **Thickening Technique Used** properties on the **General** tab of the **Plate Part Properties** dialog box. For more information, see *General Tab (Plate Part Properties Dialog Box)* (on page 127). (P4 CP:258906)

SECTION 1

Molded Forms

The Molded Forms task of Intergraph Smart™ 3D is used to define basic structure of the model, including planar plates, non-planar plates, major openings, stiffeners, and design seams. Using this task, you create:

- High-level plate and profile design systems. These systems contain properties such as type, thickness, continuity, and molded conventions.
- Child plate and profile design systems. All high-level systems have at least one child. If a high-level system has been split, it has two or more child systems.
- Root parts. These parts are created automatically as children of systems. They have thickness and inherit the properties of their parent systems. These properties are used to detail the root parts in the Structural Detailing task and to generate data for manufacturing in the Structural Manufacturing task.
- Logical connections. These connections are created automatically as children of systems and contain the connection properties between systems. These properties are used to detail the connections in the Structural Detailing task and to generate data for manufacturing in the Structural Manufacturing task.

You can start the Molded Forms task by clicking **Tasks > Molded Forms**. The Molded Forms task has these commands:



Select - Used to select objects in the model. For more information, see *Selecting Objects* (on page 40).



Planar Plate System - Places a planar plate system. For more information, see *Planar Plate System* (on page 99).



Imported Plate System - Imports an ACIS or IGES file into the model. The file is used to create a plate system, typically a complex, non-planar system such as a ship hull. For more information, see *Imported Plate System* (on page 130).



Linear Extruded Plate System - Extrudes a plate system from a 2D curve normal to the sketching plane. For more information, see *Linear Extruded Plate System* (on page 156).



Nonlinear Extruded Plate System - Extrudes a plate system from a 2D curve along a linear or nonlinear curve that you specify. For more information, see *Nonlinear Extruded Plate System* (on page 180).



Revolved Plate System - Places a plate system by rotating a curve around an axis. For more information, see *Revolved Plate System* (on page 205).



Bracket Plate System - Creates plate systems that represent primary structural brackets. Bracket orientation is defined by a plane or bracket supports. For more information, see *Bracket Plate System* (on page 227).



Advanced Plate System - Creates specialized plate systems such as side plates, ring plates, transition plates, continuity plates, and buckling plates. For more information, see *Advanced Plate System* (on page 275).



Child Plate System - Creates a plate system that is a child to another plate system. For more information, see *Child Plate System* (on page 426).



Ruled Plate System - Creates a plate system that is interpolated between two non-planar curves. For more information, see *Ruled Plate System* (on page 449).



Openings - Places a major opening in a plate system. For more information, see *Openings* (on page 475).



Member Openings - Places openings (holes) in slabs and linear member systems. For more information, see *Place Opening* (on page 705).



Execute Split - Automatically creates intersection seams and splits systems by intersection and design seams. The split creates new subsystems. For more information, see *Execute Split* (on page 891).



Manual Split - Allows multiple seams and splits to be placed. For more information, see *Manual Split* (on page 895).



Place Split - Divides a member system into multiple member parts while maintaining the design intent of the original member system. For more information, see *Place Splits* (on page 921).



Split Seam - Splits a seam by intersecting seams. For more information, see *Split Seam* (on page 897).



Split Root System - Splits a top-level (or root) plate or profile system into multiple root systems. For more information, see *Split Root System* (on page 899).



Stiffener Profile System by Intersection - Creates a stiffener profile system on the selected plate system by defining the landing curve of the stiffener as the intersection of the selected plate system and a grid plane. For more information, see *Stiffener Profile System by Intersection* (on page 723).



Stiffener Profile System by Offset - Creates a stiffener profile system on the selected plate system by defining the landing curve as being offset from an existing curve (a seam, plate boundary, plate edge, the landing curve of another profile system, or grid plane). For more information, see *Stiffener Profile System by Offset* (on page 739).



Stiffener Profile System by Projection - Creates a stiffener profile system on the selected plate system by projecting a 2D landing curve that you have sketched onto the plate system. For more information, see *Stiffener Profile System by Projection* (on page 746).



Stiffener Profile System by Table - Creates a stiffener profile system on the selected plate system by defining the landing curve as coordinates in a table. For more information, see *Stiffener Profile System by Table* (on page 755).



Tripping Stiffener System - Creates a tripping stiffener on a selected plate system. For more information, see *Tripping Stiffener System* (on page 765).



Profile Edge Reinforcement - Places a profile system along the free edge of a plate system. For more information, see *Profile Edge Reinforcement* (on page 771).



Place Linear Member - Places columns, beams, braces, and other linear members in the model. For more information, see *Place Linear Member Systems* (on page 787).



Place Can - Places a can in the model. A can is a reinforcing connection piece on a member to increase the surface area available for connections or to provide increased wall thickness or both. For more information, see *Place Can* (on page 860).



Logical Connection - Creates a relationship between two molded form systems to identify that the two systems are connected. For more information, see *Logical Connection* (on page 943).



Place Assembly Connection - Places assembly connections between linear member systems. For more information, see *Place Assembly Connection* (on page 952).



Place Member Generic Assembly Connection - Places an assembly connection between a built-up member part and an intersecting part. For more information, see *Place Member Generic Assembly Connection* (on page 968).



Seam by Intersection - Creates a seam on the selected plate system by defining the landing curve of the seam as the intersection of the selected plate system and a grid plane. For more information, see *Seam by Intersection* (on page 902).



Seam by Offset - Creates a seam on the selected plate system by defining the landing curve as being offset from an existing curve (such as another seam, plate boundary, profile system, or grid plane). For more information, see *Seam by Offset* (on page 905).



Seam by Projection - Creates a seam on the selected plate system by projecting a 2D or 3D landing curve that you have sketched onto the plate system. For more information, see *Seam by Projection* (on page 908).



Seam by Table - Creates a seam on the selected plate system by defining the landing curve as coordinates in a table. For more information, see *Seam by Table* (on page 912).



Profile Seam - Creates a design seam on a profile or on all profiles that belong to a plate system. For more information, see *Profile Seam* (on page 914).



Profile Knuckle - Places a manual profile knuckle on a selected profile system. For more information, see *Profile Knuckle* (on page 935).



Reference Curve by Intersection - Creates a reference curve on the selected plate system by defining the landing curve of the reference plane as the intersection of the selected plate system and a grid plane. For more information, see *Reference Curve by Intersection* (on page 973).



Reference Curve by Offset - Creates a reference curve on the selected plate system by defining the landing curve as being offset from an existing curve (such as another grid, mark, or reference). For more information, see *Reference Curve by Offset* (on page 976).



Reference Curve by 2D Sketch - Creates a reference curve on the selected plate system by projecting a 2D landing curve that you have sketched onto the plate system. For more information, see *Reference Curve by 2D Sketch* (on page 980).



Reference Curve from External File - Creates and modifies knuckle reference curves. These curves require that the edges of the sheetbody faces carry an attribute that reflects that they are knuckles and relates them to the appropriate reference curve. For more information, see *Reference Curve from External File* (on page 984).



Execute Detailing - Creates detail parts from the root part of a plate or profile system. For more information, see *Execute Detailing Command* (on page 989).

Import IGES - Imports 3D geometric data from a file in the IGES (Initial Graphics Exchange Standard) format. This command is on the **File** menu. For more information, see *Import IGES (File Menu)* (on page 1216).

Import ACIS - Imports 3D geometric data from a file in the ACIS .sat format. This command is on the **File** menu. For more information, see *Import ACIS (File Menu)* (on page 1216).

Import Stiffeners - Imports stiffener profile systems from data in a Microsoft Excel workbook. This command is on the **File** menu. For more information, see *Import Stiffeners (File Menu)* (on page 1217).

Import Seams - Imports design seams from data in a Microsoft Excel workbook. This command is on the **File** menu. For more information, see *Import Seams (File Menu)* (on page 1225).

Import Ship Structure - Imports Tribon ship structure data for use as background structure in planning and outfitting work. This command is only available if the Smart 3D Tribon Interface is installed. For more information, see Import Ship Structure (File Menu).

Export IGES - Exports 3D geometric data to a file in the IGES format. This command is on the **File** menu. For more information, see *Export IGES (File Menu)* (on page 1236).

Export ACIS - Exports geometric data to a file in the ACIS .sat format. This command is on the **File** menu. For more information, see *Export ACIS (File Menu)* (on page 1236).

Export Stiffeners - Exports stiffener data to a Microsoft Excel workbook. This command is on the **File** menu. For more information, see *Export Stiffeners (File Menu)* (on page 1237).

Export Seams - Exports seam data to a Microsoft Excel workbook. This command is on the **File** menu. For more information, see *Export Seams (File Menu)* (on page 1239).

Neutral/GeniE Export - Exports model data to a neutral or GeniE XML format. This command is on the **File** menu. For more information, see *Export Neutral/GeniE XML (File Menu)* (on page 1241).

Copy by Family - Replicates objects in the model maintaining a relationship between the copies. This command is available on the **Edit** menu. For more information, see *Copy by Family* (on page 1255).

Copy by Family Synchronize - Opens the **Copy by Family** ribbon with the last used family set as the default family. This command is available on the **Edit** menu. For more information, see *Copy By Family Synchronize* (on page 1315).

Copy by Family Audit Report - Creates a report that allows you to verify that the source has been copied to the target. This command is available on the **Tools** menu. For more information, see *Copy By Family Audit Report* (on page 1316).

Change Geometry Type - Modifies the geometry type of a plate. This command is available on the **Edit** menu. For more information, see *Change Geometry Type (Edit Menu)* (on page 1385).

Topological Points - Creates associative points directly in the graphic view. This command is available on the **Insert** menu. For more information, see *Topological Points (Insert Menu)* (on page 1323).

Delay Settings - Delays the update of existing Structural Detailing and Structural Manufacturing objects when the parent Molded Form object is modified, putting the objects on the **To Do List** for future update. This command is available on the **Tools** menu. For more information, see *Delay Settings (Tools Menu)* (on page 1366).

Data Integrity Check - Verifies the data integrity of Molded Forms systems in your model before creating detailed parts in the Structural Detailing task. This command is available on the **Tools** menu. For more information, see *Data Integrity Check (Tools Menu)* (on page 1371).

Measure Explicit - Measures girth distances along landing curves in a cumulative fashion. This command is available on the **Tools** menu. For more information, see *Measure Explicit (Tools Menu)* (on page 1386).

Automatic Rebound - Changes the boundary from one object to another. This command is available on the **Tools** menu. For more information, see *Rebound objects* (on page 1391).

Submit Batch Job - Runs batch processes using the Intergraph Batch Services framework. This command is available on the **Tools** menu. For more information, see *Submit Batch Job* (on page 1380).

Un-Split - Merges plates into a single plate leaving the seam in the model so that you can use the seam to split the plate again at a later time. For more information, see *Un-Split* (on page 895).

Tripping Bracket - Provides compatibility with brackets created with **Bracket Plate System** in earlier versions of the software. For more information, see *Tripping Bracket* (on page 1425).

★ IMPORTANT To create new tripping brackets, use **Bracket Plate System** and

select **Plane by Selected Supports**  as the value for **Select Plane**.

Shell Plate Unfold Preview - Generates a graphic preview of an unfolded plate shape with frame marks, roll line marks, surrounding minimum rectangle and annotation representing dimensions of minimum rectangle, unfolded surface area and percentage of material utilization in a private graphic view. For more information, see *Shell Plate Unfold Preview* (on page 1449).

See Also

Molded Forms Workflow (on page 28)

SECTION 2

Molded Forms Workflow

All molded forms objects are placed in the model using information defined in the reference data. The first steps are to review, edit, and otherwise customize the delivered reference data. These steps are performed by an administrator with permissions to change the reference data prior to the start of modeling.

After the reference data is customized, your administrator needs to define the molded forms systems in the Systems and Specifications task. The systems are typically in a predefined work breakdown structure.

Because molded forms rely on grid planes, your administrator must go to the Grids task and place grid planes before you can place any molded forms objects in your model.

After the reference data and the needed systems are defined, you can begin placing molded forms objects in your model.

Place the Hull

Import an ACIS or IGES file into your model to create the hull. For more information, see *Import a plate system* (on page 136).

Place Decks and Bulkheads

Create the major plate systems in your model that represent the decks and bulkheads. You have several methods for placing these plate systems:

Place a planar plate system (on page 104)
Place a linear extruded plate system (on page 161)
Place a nonlinear extruded plate system (on page 185)
Place a revolved plate system (on page 209)
Create a ruled plate system using existing curves (on page 454)
Place a custom plate system using geometric constructions (on page 282)
Bracket Plate System (on page 227)

Place Openings

After the deck and bulkheads are in the model, add the major openings such as cargo hold hatches.

Place an opening by sketching (on page 479)
Place an advanced opening (on page 480)

Create Stiffener Profile Systems

Place stiffeners as needed. You have several methods for placing stiffeners:

Place stiffener profile by intersection (on page 728)
Place stiffener profile by offset (on page 745)
Place stiffener profile by projection (on page 753)
Place stiffener profile by table (on page 760)

Place tripping stiffener (on page 769)
Import stiffener or seam data (on page 1217)

Place Seams

Place design and planning seams as needed. You have several methods for placing seams:

Seam by Intersection (on page 902)
Seam by Offset (on page 905)
Seam by Projection (on page 908)
Seam by Table (on page 912)
Profile Seam (on page 914)
Import stiffener or seam data (on page 1217)

Place Edge Reinforcement Profile Systems

Place edge reinforcements as needed. For more information, see *Place edge reinforcement profile* (on page 774).

Execute the Splits

When all the design seams are placed, you need to split the plate and profile systems.

Split a plate system (on page 893)
Split a profile or member system (on page 893)
Create multiple intersections seams (on page 895)
Split a seam by intersecting seams (on page 897)
Split a member that intersects another member (on page 925)

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Marine Structure Hierarchy in the Workspace Explorer

Plates and profiles gain children objects in the **Workspace Explorer** hierarchy as the model progresses from early design in Molded Forms through manufacturing parts in Structural Manufacturing. The following objects appear on the **System** tab of the **Workspace Explorer**:

Root system

Contains the top-level properties for a plate or profile, such as continuity, tightness, material grade, and molded conventions, but does not contain all physical attributes. For example, a root plate system is a surface without thickness, and a root profile system is a landing curve.

Leaf or **child system**

Inherits the properties of the parent root system. Properties can be manually changed to be different from the root system. Each root system is created with a leaf system. When the root system is split, additional leaf systems are created.

Plate part

Inherits the properties of the parent leaf system and begins to carry physical attributes. Each leaf system has a plate part. There are two types of plate parts:

Light part

The initial part created in Molded Forms. Plate parts have thickness and profile parts have cross-sections, but other physical attributes are not considered. For example, parts are not trimmed at their boundaries and profile part locations are not adjusted based on plate part thickness.

Detailed part

Replaces the light part after detailing the part in Structural Detailing. Parts gain additional physical attributes. For example, plates are now trimmed at their boundaries; profile webs and flanges are trimmed.

Root logical connection

Contains top-level properties for a connection between plate and profile systems. Each root logical connection is a child of a root system.

Leaf or **child logical connection**

Each root logical connection is created with a leaf logical connection. When the root logical connection is split, additional leaf logical connections are created.

Assembly connection

Inherits the properties of the parent leaf logical connection. An assembly connection is created when parts are detailed in Structural Detailing. Each assembly connection is a child of a leaf logical connection. An assembly connection contains properties used for physical connections (welds) and features such as end cuts, slots, and collars.

Feature

Contains properties and physical attributes for detailed features such as end cuts, slots, or collars. A feature is created when parts are detailed in Structural Detailing. Each feature is a child of an assembly connection.

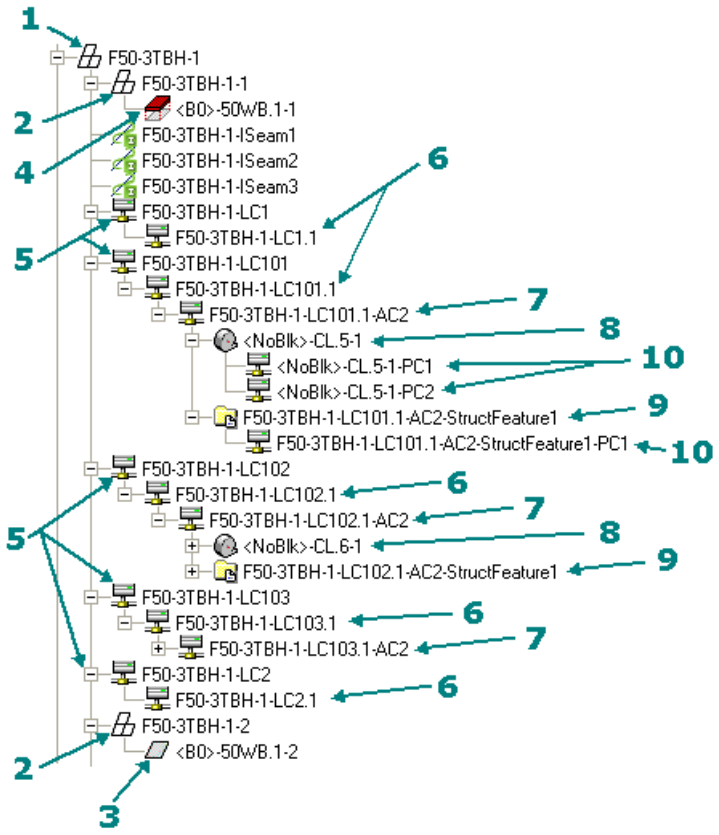
Physical connection

Contains properties and physical attributes for welds. Physical connections are created for

each assembly connection when parts are detailed in Structural Detailing. End-to-end and edge-to-edge connections have one physical connection for each part. Each physical connection is a child of an assembly connection or a feature.

Examples

Plate



1 - Root system

2 - Leaf system

3 - Light part

4 - Detailed part

5 - Root logical connection

6 - Leaf logical connection

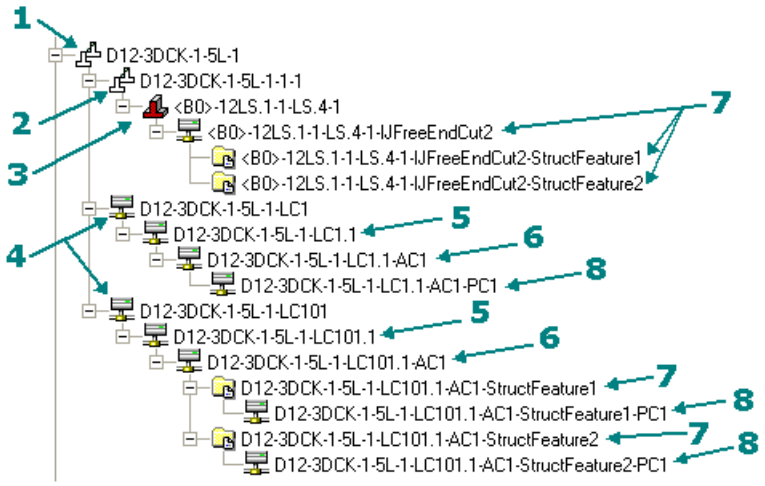
7 - Assembly connection

8 - Collar feature

9 - Slot feature

10 - Physical connection

Profile



1 - Root system

2 - Leaf system

3 - Detailed part

4 - Root logical connection

5 - Leaf logical connection

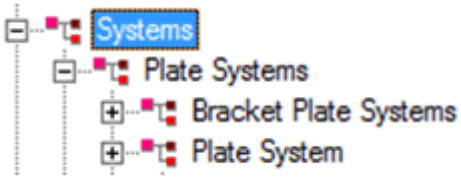
6 - Assembly connection

7 - End cut feature

8 - Physical connection

NOTES

- Objects created in Structural Manufacturing, such as manufacturing parts, templates, and pin jigs, are children to parts, blocks, or assemblies on the **Assembly** tab of the **Workspace Explorer**.
- **Plate Systems** display at the same level in the hierarchy as **Bracket Plate Systems**. If you filter on **Plate Systems**, the software also displays **Bracket Plate Systems** as shown in the following figure. This happens because the leaf system of a bracket plate system is a plate system object. The bracket plate system is included in the workspace because it is the parent of the leaf plate system.



Icons in the Workspace Explorer

 Analysis Folder

 Analysis Model

 Assembly

 Assembly Connection

 Beam Part - undetailed

 Beam Part - detailed

 Beam Part - detailed, manufactured

 Bearing Plate - undetailed

 Bearing Plate - detailed

 Bearing Plate - detailed, manufactured

 Bearing Plate - group member

 Bearing Plate - group member, manufactured

 Bearing Plate - group master

 Bearing Plate - group master, manufactured

 Block

 Boundary Condition

 Bracket System or Leaf System

 Bracket Part - undetailed

 Bracket Part - detailed

 Bracket Part - detailed, manufactured

 Bracket Part - group member

 Bracket Part - group member, manufactured

 Bracket Part - group master

 Bracket Part - group master, manufactured

 Cable Nozzle on Equipment

 Cabletray Nozzle on Equipment

 Collar Plate - undetailed

 Collar Plate - detailed

 Collar Plate - detailed, manufactured

 Collar Plate - group member

 Collar Plate - group member, manufactured

 Collar Plate - group master

 Collar Plate - group master, manufactured

 Common Part Manager

 Conduit Nozzle on Equipment

 Coordinate System

 Coordinate System Axis

 Coordinate System Grid Plane

 Coordinate System Elevation Plane

 Coordinate System Radial Plane

 Coordinate System Radial Cylinder

 Curved Member Part - undetailed

 Curved Member Part - light

 Curved Member Part - light, manufactured

 Curved Member Part - detailed

 Curved Member Part - detailed, manufactured

 Curved Member Part - group member

 Curved Member Part - group member, manufactured

 Curved Member Part - group master

 Curved Member Part - group master, manufactured

 Designed Equipment Component

 Designed Equipment

 Designed Handrail

 Designed Member - undetailed

 Designed Member - light

 Designed Member - light, manufactured

 Designed Member - detailed

 Designed Member - detailed, manufactured

 Designed Member - group member

 Designed Member - group member, manufactured

 Designed Member - group master

 Designed Member - group master, manufactured

 Edge Treatment

 Equipment

 Equipment Component

 Equipment Foundation

 Equipment Shape

 Equipment Solid

 Equipment Solid - Added

 Equipment Solid - Subtracted

 Equipment Solid - Suppressed

 Feature (includes corner feature, edge feature, face feature, slot, and so on)

 Footing

 Footing Components

 Foundation Port

 Frame Connection

 Handrail

 HVAC Nozzle on equipment

 Insert Plate - undetailed

 Insert Plate - detailed

 Insert Plate - detailed, manufactured

 Insert Plate - group member

 Insert Plate - group member, manufactured

 Insert Plate - group master

 Insert Plate - group master, manufactured

 Ladder

 Linear Member Part - undetailed

 Linear Member Part - light

 Linear Member Part - light, manufactured

 Linear Member Part - detailed

 Linear Member Part - detailed, manufactured

 Linear Member Part - group member

 Linear Member Part - group member, manufactured

 Linear Member Part - group master

 Linear Member Part - group master, manufactured

 Load Case

 Load Combination

 Member Assembly Connection Cutback Feature

 Member Assembly Connection Plate Part

 Member Assembly Connection

 Member Fireproofing

 Member Part

 Member Split Connection

 Member System

 Opening

 Panel - manufactured

 Physical Connection

 Pipe Nozzle on Equipment

 Planning Joint - butt weld

 Planning Joint - lap weld

 Planning Joint - tee weld

 Planning Joint Folder

 Plate - light

 Plate Part - undetailed

 Plate Part - light

 Plate Part - light, manufactured

 Plate Part - detailed

 Plate Part - detailed, manufactured

 Plate Part - detailed, manufactured, with manufacturing part assembly child

 Plate Part - group member

 Plate Part - group member, manufactured

 Plate Part - group master

 Plate Part - group master, manufactured

 Plate System or leaf system

 Profile Edge Reinforcement System or Leaf System

 Profile Edge Reinforcement Part

 Profile Edge Reinforcement Part - light

 Profile Edge Reinforcement Part - light, manufactured

 Profile Edge Reinforcement Part - detailed

 Profile Edge Reinforcement Part detailed, manufactured

 Profile Edge Reinforcement Part - group member

 Profile Edge Reinforcement Part - group member, manufactured

 Profile Edge Reinforcement Part - group master

 Profile Edge Reinforcement Part - group master, manufactured

 Seam - Design

 Seam - Intersection

 Seam - Planning

 Seam - Straking

 Slab Assembly Connection

 Slab

 Stair

 Standalone Beam Part - undetailed

 Standalone Beam Part - detailed

 Standalone Beam Part - detailed, manufactured

 Standalone Plate Part - undetailed

 Standalone Plate Part - detailed

 Standalone Plate Part - detailed, manufactured

 Standalone Plate Part - group member

 Standalone Plate Part - group member, manufactured

 Standalone Plate Part - group master

 Standalone Plate Part - group master, manufactured

 Standalone Profile Edge Reinforcement Part - undetailed

 Standalone Profile Edge Reinforcement Part - detailed

 Standalone Profile Edge Reinforcement Part - detailed, manufactured

 Standalone Profile Edge Reinforcement Part - group member

 Standalone Profile Edge Reinforcement Part - group member, manufactured

 Standalone Profile Edge Reinforcement Part - group master

 Standalone Profile Edge Reinforcement Part - group master, manufactured

 Standalone Stiffener Part - undetailed

 Standalone Stiffener Part - detailed

 Standalone Stiffener Part - detailed, manufactured

 Standalone Stiffener Part - group member

 Standalone Stiffener Part - group member, manufactured

 Standalone Stiffener Part - group master

 Standalone Stiffener Part - group master, manufactured

 Stiffener Part - undetailed

 Stiffener Part - light

 Stiffener Part - light, manufactured

 Stiffener Part - detailed

 Stiffener Part - detailed, manufactured

 Stiffener Part - group member

 Stiffener Part - group member, manufactured

 Stiffener Part - group master

 Stiffener Part - group master, manufactured

 Stiffener System or Leaf System

 Wall Assembly Connection

 Wall Part

 Wall Run

 Wall System

Selecting Objects

All objects in the **Molded Forms** task have properties that you can edit. Using the **Select**  command on the vertical toolbar, you select the object that you want to edit or work with.



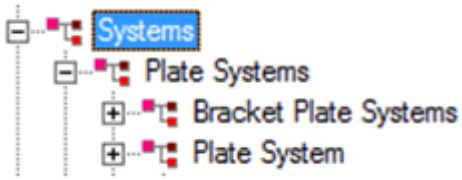
An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains pre-defined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered objects in a graphic view and in the **Workspace Explorer**. For example, if you select **Seams**, you can select only design, intersection, planning, or a seam point in a graphic view or in the **Workspace Explorer**.

The **Molded Forms** task includes these filters:

Bracket Plate Systems

Limits your selection to bracket plate systems.

NOTE **Plate Systems** display at the same level in the hierarchy as **Bracket Plate Systems**. If you filter on **Plate Systems**, the software also displays **Bracket Plate Systems** as shown in the following figure. This happens because the leaf system of a bracket plate system is a plate system object. The bracket plate system is included in the workspace because it is the parent of the leaf plate system.



Connections

Limits your selection to:

- Indent 2
  - Logical connections
  - Frame connections
  - Split connections
    - Indent bullet 3
      - Indent bullet 4

Control Points

Limits your selection to control points. You define the location of a control point with **Insert** > **Control Point**.

Construction Graphics

Limits your selection to lines, line strings, arcs, circles, and rectangles created with **Insert** > **Construction Graphics**.

Members

Limits your selection to member parts.

Member Systems

Limits your selection to:

- Standard members
- Design members
- Cans

Molded Forms (Leaf Level)

Limits your selection to:

- Leaf plate systems
- Leaf stiffener systems
- Leaf edge reinforcements
- Leaf logical connections
- Grids

- Standard members
- Cans

Molded Forms (Root Level)

Limits your selection to:

- Root plate systems
- Root bracket plate system
- Root stiffener systems
- Root edge reinforcements
- Root beam systems
- Root logical connections
- Seams
- Seam point
- Openings
- Knuckles
- Reference curves
- Grids
- Frames (Child of grid frame system)
- Frame connections
- Split connections
- Standard members
- Designed members
- Cans

Openings

Limits your selection to major openings and member openings.

Parts

Limits your selection to:

- Plate parts
- Profile parts
- Beam parts
- Edge reinforcement parts
- Member parts

Points

Limits your selection to topological points. You define the location of a control point with the **Insert** > **Topological Points** command.

RefCurve on Surface & Knuckles

Limits your selection to:

- Reference curves
- Plate knuckles
- Profile knuckles

Root Plates

Limits your selection to:

- Root plate systems
- Leaf plate systems
- Designed members
- Cans

Root Profiles

Limits your selection to:

- Root stiffener systems
- Leaf stiffener systems
- Root edge reinforcement systems
- Leaf edge reinforcements systems
- Standard members
- Design members

Seams

Limits your selection to:

- Design seams
- Intersection seams
- Planning seams
- Seam points

All

Allows you to select any object, even objects created in another task.

More

Allows you to create and select customized filters. For more information, see *Select Filter Dialog Box* (on page 1282) in the *Common User's Guide*.

 **Inside Fence**

Allows you to select all objects entirely inside the fence.

 **Overlapping Fence**

Allows you to select all objects entirely inside the fence and those objects outside but touching the fence at some point.

Direction and Orientation Glyphs for Objects

The software provides feedback on the direction, orientation, and rotation of objects using glyphs. A glyph is a straight or curved line with a cone-shaped arrow head that indicates a particular direction. When you are creating or modifying objects, glyphs assist you so that you can properly set design parameters like twist, mounting angle, and offset.

The software displays glyphs for:

- Root plate systems and leaf plate systems
- Root stiffener systems and leaf stiffener systems
- Seams
- Reference curves

Glyphs display using the defined handle color as defined in **Tools > Options**.

Glyphs for Plate Systems

Glyphs for plate systems display at the approximate center of plate as a straight line with two arrows. The top arrow represents the thickness direction, and the bottom arrow represents normal direction. For example:

- Glyphs for plate systems with both thickness and normal pointing in the same direction



- Glyphs for plate systems with thickness and normal pointing in opposite directions



Glyphs for Bracket Systems

Glyphs for bracket systems display at the bracket origin for tripping brackets and at the bracket reference position for brackets by plane. The three glyphs represent the thickness direction, the normal direction, and the U-direction vector.

Glyphs for Profile Systems

Glyphs for profile systems display at the start point of the profile landing curve. The three glyphs represent the profile direction, primary orientation, and angular orientation.



The profile direction glyph represents the tangent of the profile at the start point. The angular orientation glyph represents the direction of profile orientation when a positive angle is applied for twist.

Glyphs for Seams and Reference Curves

Glyphs for seams and reference curves display at the start point of the landing curve and display the direction of the landing curve.



See Also

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Permission Group Behaviors Between Tasks

Permission groups provide controls for when, where, and by whom the model can be updated. Permission groups are assigned according to several common types of workflow. They typically fall into the following categories:

- By Function: such as Outfitting versus Structure, or Piping versus Equipment versus Molded Forms versus Structural Detailing.
- By System: such as Hull versus Deck1 versus TransvBhd1. This includes the ability to place the hull system (and its child leaf systems, seams, connections related seams, reference curves and openings) in a different permission group than the hull detailing objects and other connected objects.
- By Area: Fwd versus Aft, or by block or assembly.
- Combinations: for example, By Function and By Area (Molded Forms for Block1 versus Detailing for Block1).

The permission group of an object is not modified regardless of the **Active Permission Group** unless the permission group is explicitly changed on the **Configuration** tab of the **Properties** dialog box as described under the change propagation sections below.

Molded Forms Behavior

Plate systems and profile systems are assigned to the **Active Permission Group**.

Leaf systems, light parts, reference curves, and logical connections are assigned the permission group of their parent.

Seams and openings are assigned to the permission group of the plate or profile system they cut. You must have write access to the root plate system being cut when placing an opening or design seam.

In the **Execute Split** command, seams and knuckles that you do not have permission to modify are not selectable in the table.

Logical connections between profile systems and plate systems are in same permission group as the profile systems.

Logical connections between different root systems are created as children of the bounded or penetrated system.

Summary of Molded Forms permission behaviors:

Object	Permission Group	Comments
Root plate system	Active Permission Group	
Leaf plate system	Same as root plate system	
Root and child logical connections	Same as dependent root plate system	
Reference curve	Same as root plate system	
Design seam, intersection seam	Same as root plate system	Must have write access to root plate system
Opening	Same as root plate system	Must have write access to root plate system
Manual logical connection	Permission group of system for which you have write access, use first system selected if you have write access to both systems	Must have write access to at least one of the systems being connected
Root profile system	Active Permission Group	Profiles are handled similarly to plates
Logical connection between stiffener or edge reinforcement and the plate being stiffened	Same permission group as root profile system	
Beam	Active Permission Group	

Molded Forms Change Propagation

The root system and its child leaf system, logical connections, reference curves, seams and openings always have the same permission group. When the permission group is changed for any of these objects, the permission group is also changed for the other objects.

If plate system A is bounded by plate system B, you can modify A without write access to B. Plate System B is placed on the **To Do List** as **Out of Date** if the change to A impacts B.

Structural Detailing Behavior

The detailed part permission group is not assigned from the parent system permission group. Detailed parts created with **Execute Detailing** are assigned to the **Active Permission Group**. You can modify this assignment. You must have write permission to the part to detail the part.

If a detailed part is deleted, the resulting light part maintains the permission group of the detailed part.

You can assign different permission groups to parts sharing the same leaf system as created by a planning seam.

You can only add seams and features to a part if you have write permissions to the part.

Summary of Structural Detailing permission behaviors:

Object	**Permission Group**	**Comments**
Light Root Plate Part (before detailing)	Same as plate system at creation	
Detailed Root Plate Part	Active Permission Group when detailed. Part keeps this permission group if the detail part is deleted.	Must have write access to the light part
Leaf part	Same as root part	
Straking seam	Same as root part	Must have write access to root part
Feature placed by command (edge, corner, sketched, free edge treatment)	Same as root part	Must have write access to root part
Assembly Connection	Same as dependent part	
Feature, part, physical connection created by assembly connection	Same as assembly connection	
Free end cut object and its features	Same as root part	
Standalone part	Active Permission Group	

Structural Detailing Change Propagation

The detailed part, its child leaf parts, features and straking seams always have the same permission group. Assembly connections always have the same permission group as the dependent part in the connection. Assembly connection children always have the same permission group as the assembly connection. Standalone parts plus associated features and connections always have the same permission group. When the permission group is changed for any of these objects, the permission group is also changed for the other objects.

If you do not have write access to a part, the part is placed on the **To Do List** as **Out of Date** if a change is made to the part. Assembly connections and features are also not created until the part is updated on the **To Do List** by someone with write access.

Hole Management Behavior

Hole traces and outfitting catalog hole fittings are assigned to the **Active Permission Group**.

Hole cuts and structural hole fittings are assigned the permission group of their parent structural part (plate or profile).

Summary of Hole Management permission behaviors:

Object	**Permission Group**	**Comments**
Hole trace	Active Permission Group	
Hole cut	Same as root plate part	Must have write access to parent plate part. Root plate part must be detailed in the Structural Detailing task.
Outfitting catalog hole fitting	Active Permission Group	
Structural hole fitting	Active Permission Group	Must have write access to parent plate part. Root plate part must be detailed in the Structural Detailing task.

Hole Management Change Propagation

The permission group of an outfitting catalog hole fitting can be changed without changing the permission group of the hole trace.

The permission group of a structural hole fitting can be changed without changing the permission group of the hole cut or parent plate part.

A hole cut and its parent plate part always have the same permission group. When the permission group is changed for one of these objects, the permission group is also changed for the other object.

Planning Behavior

Planning permission group behavior is related to Molded Forms and Structural Detailing behavior because planning seams split leaf systems.

When **Manage Intersections** is run and an intersecting part is set to **Split** or **Offset**, planning seams are created in the **Active Permission Group**. You must have write access to the root system.

Summary of Planning permission behaviors:

Object	Permission Group	Comments
Planning seam	Active Permission Group	Must have write access to the root system of the parts being split.
Block or assembly	Active Permission Group	

Planning Change Propagation

Write access is required to both parts to remove a planning split using **Flatten Assembly** or **Manage Intersections**. The resulting new part is assigned to the same permission group as the original parts if they belonged to the same permission group. The new part is assigned to the **Active Permission Group** if the original parts were in different permission groups.

Global Workshare Behavior

Global Workshare allows managed replication of the model at several sites. One of the restrictions of a Workshare configuration is that you only have Read access to objects that are controlled by non-local permission groups.

A system is a logical grouping of sub-systems. When you add or remove a sub-system, you are modifying the definition of the parent system. Therefore, you must have write access to the parent system. You do not need write access to the grandparent system.

When participating in a Global Workshare Configuration, you must manage all permission groups at the host site. The sub-system requirement to have write access to its parent system is not possible if the sub-system's permission group is created at the satellite site and the parent system's permission group is created at the host site.

For example, your host site is Houston and your satellite site is London. You create a system named "Midbody", and its controlling permission group is in Houston. You assign John, who works in London, write access. During the workshare replication process, the Midbody system and permission group is replicated in London. In London, John can add systems, plates, profiles and whatever else he wants to the Midbody system because John was given write access to the system's permission group in Houston. John cannot delete or change any of the properties of the Midbody system in London because the host site, Houston, owns it. He can only add objects to the system. If John were to travel to Houston and log on there, he could delete or change any of the properties of the Midbody system because the Houston host site owns it.

Best Practices

You may encounter situations where you do not know the best modeling technique. The following best practices guide you through the modeling process. Because your modeling environment is unique, specific instructions are not possible. However, enough detail is provided to help you work through the solutions.

Topics

Bracket Best Practices 50
Knuckle Best Practices 56
Landing Curve Best Practices 56
Model Bulkheads with Multiple Openings 57
Seams Best Practices 60
Session File Best Practices 64
Sketch 2D Best Practices 65
Split Best Practices 74

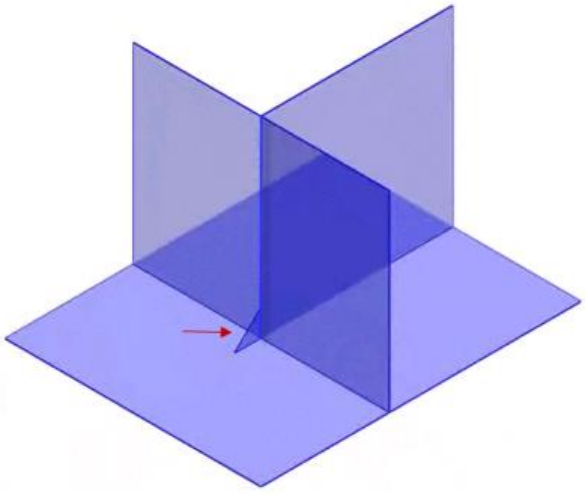
Bracket Best Practices

Bracket Plate System creates plate systems for primary structural brackets that can be stiffened, penetrated by stiffeners on the bounding structure, and split with design seams. Use the following best practices when working with the command. For more information, see *Bracket Plate System* (on page 227).

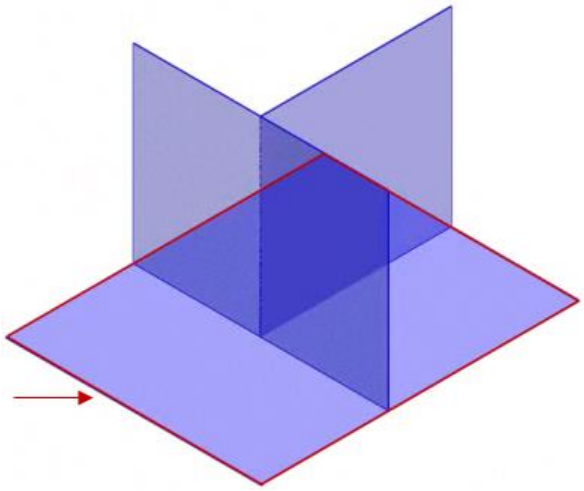
Backside support

When a support is on the backside of another plate, select the edge of the support. Selecting the system results in an error.

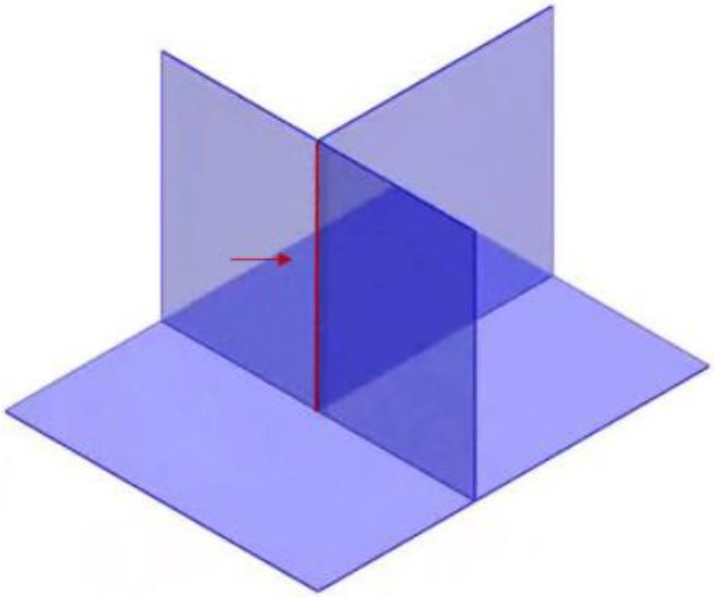
The following example shows a bracket with the backside edge of a plate as one of the supports.



1. Select the desk plate system as **Support 1**.



2. The second support is a bulkhead on the backside of another bulkhead. Using **QuickPick,** select the edge of second support as **Support 2**. Do not select the plate system.



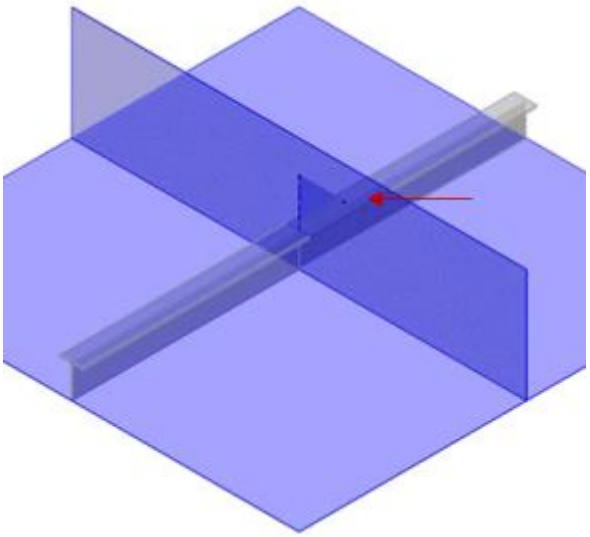
Length value and bracket parameters

When the supports are non-linear or non-planer, select a value for **Length** that is closest to or greater than bracket height and width parameters.

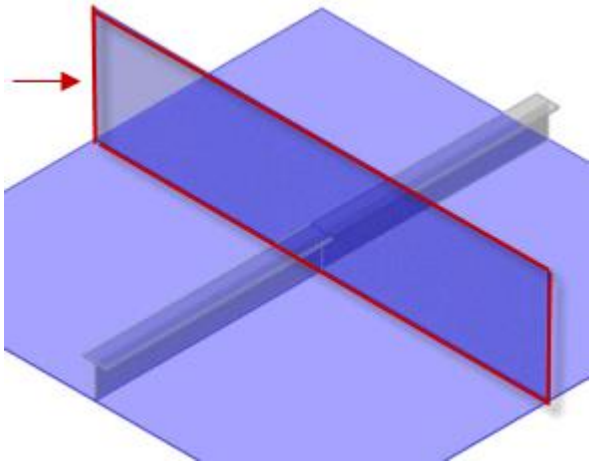
Ambiguity in a bracket support

When a bracket support has an ambiguity, select the correct side of the ambiguity.

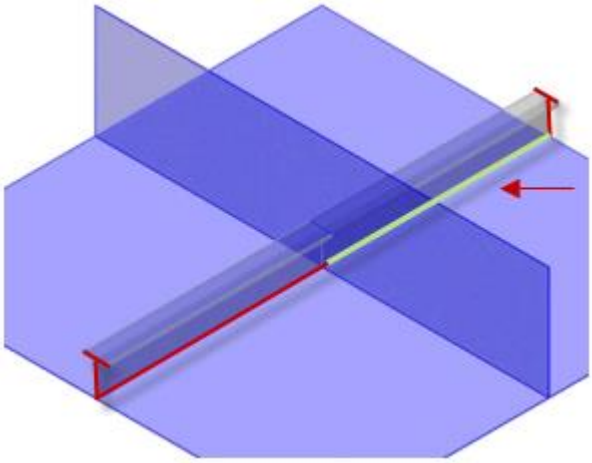
The following example shows a bracket that is on the far side of the bulkhead.



1. Select the bulkhead plate system as **Support 1**.



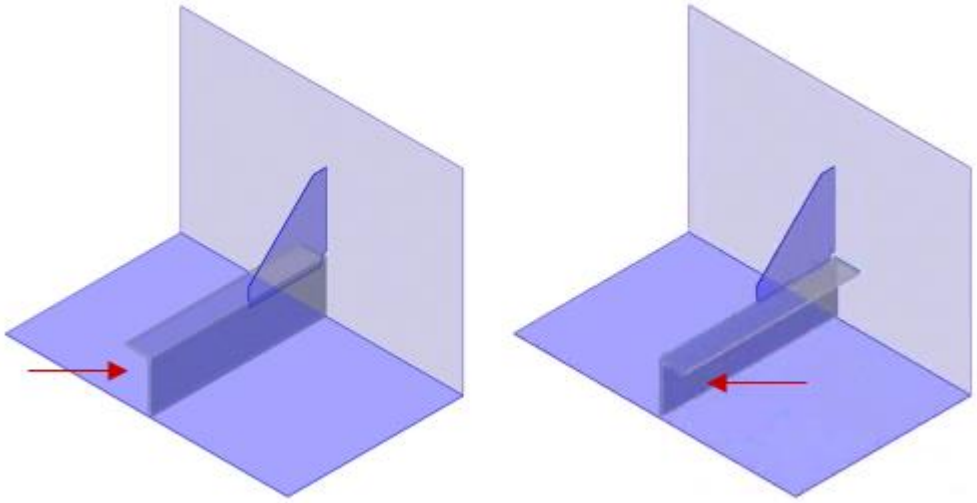
2. Select the far side of the profile system as **Support 2**.



Bracket thickness

When modifying the secondary orientation of a lapped bracket profile support, also modify the bracket thickness direction so that the bracket is properly lapped.

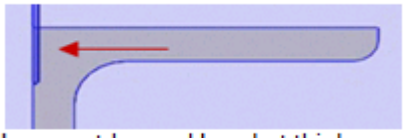
In the following example, a lapped bracket is placed on a profile system, and the secondary orientation of the profile system is modified.



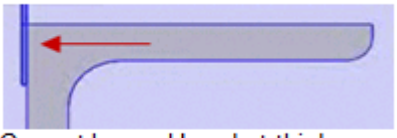
1. Select the profile system you want to modify and click **Properties** .
2. Select **Section Orientation** and modify **Secondary Orientation** as needed.
3. Click **Finish**.

   The orientation of the profile system is modified.

4. Check if the lapped bracket thickness direction is correct.



Incorrect lapped bracket thickness direction



Correct lapped bracket thickness direction

5. If the lapped bracket thickness direction is incorrect, select the bracket and click **Properties** .
6. Select **Molded Conventions** and modify the **Thickness Direction** as needed.

Molded Forms brackets versus Structural Detailing brackets

When a bracket must be penetrated, split, stiffened, or is needed for strength calculations (in an external analysis tool) or class drawings, use **Bracket Plate System** in **Molded Forms** instead of **Place Bracket Parts** in **Structural Detailing**.

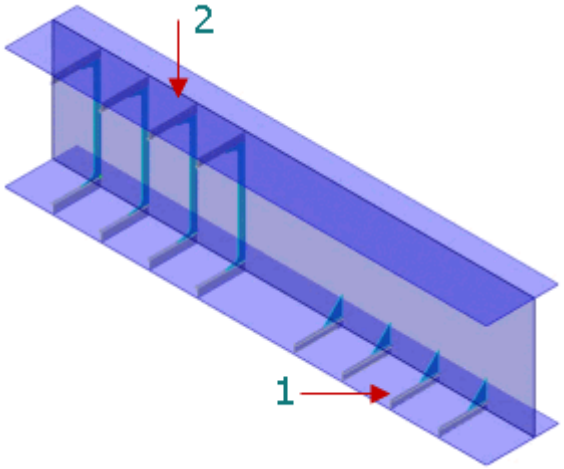
Bracket symbols

Use a bracket symbol instead of a sketched bracket when a profile is a support and minimum height of the bracket nose is important. The nose height is not maintained on a sketched bracket when the profile height is modified.

Create multiple brackets using Auto Finish Mode

When you want to create multiple brackets with the same symbol and common supports, use **Auto Finish Mode** .

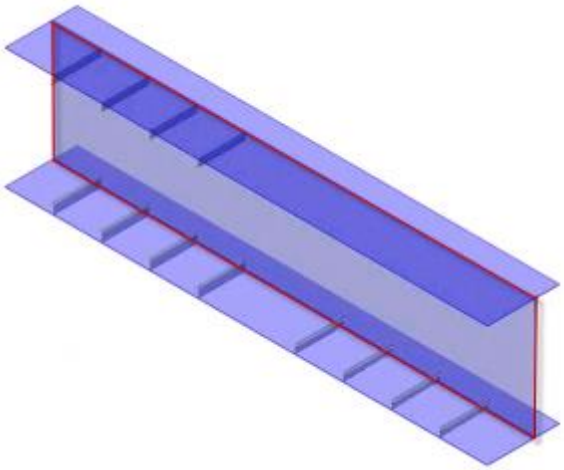
The following example shows a plate system with multiple two-support and three-support brackets created using **Auto Finish Mode** .



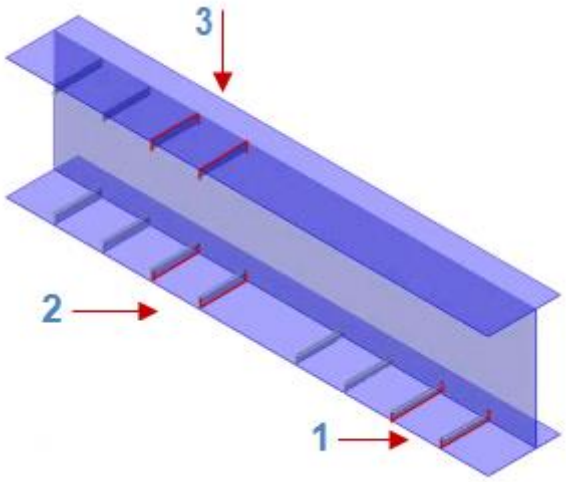
1 - Two-support bracket

2 - Three-support bracket

1. For a two-support bracket, select the plate system as **Support 1**.



2. Select **Support 2** and click **Auto Finish Mode** to create two-support brackets each time you select a similar **Support 2**.
3. For a three-support bracket, use **Bracket Shape**.
4. Select the plate system as **Support 1**.
5. Select similar **Support 2** and **Support 3** each time you want to create a three-support bracket.



1 - **Support 2** for two-support bracket

2 - **Support 2** for three-support bracket

3 - **Support 3** for three-support bracket

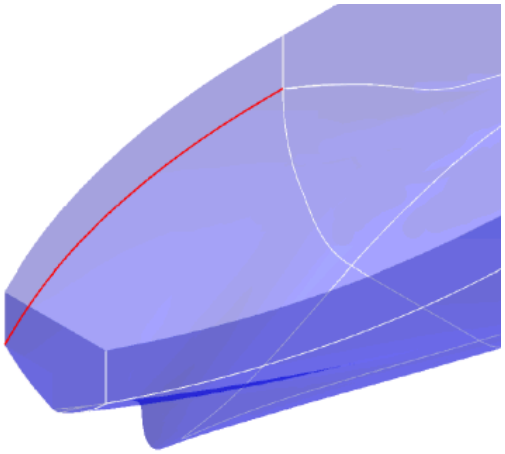
Logical connection

When the bracket supports are not loaded in the workspace, you can identify the supports by the bracket logical connections in the **Workspace Explorer**.

Knuckle Best Practices

You can use different methods to split larger plate and profile systems into smaller subsystems for detailed design and manufacturing. However, when using a knuckle instead of a split, follow these best practices.

- Do not use seams to split knuckles.
- Do not create seams too close to a knuckle curve or point. Seam creation fails if the knuckle is closer than 5mm.
- Do not delete a knuckled reference curve.
- When a knuckle reference curve transitions to a tangent, check that it also connects to another seam.



- When you have a plate bend, such as a flanged plate that is concave (thickness direction is toward the center of the radius), ensure that the radius is equal to or smaller than the plate thickness.

See Also

Placing Plate Systems (on page 96)
Placing Profile Systems (on page 721)
Creating Seams (on page 888)
Creating Reference Curves (on page 972)

Landing Curve Best Practices

Landing curves are defined when creating profile systems, seams, and reference curves by projection. When creating landing curves, use the following best practices. For more information, see *Landing Curve Methods* (on page 87).

- Avoid multi-lump (non-continuous) landing curves. For example, profiles crossing openings or openings cutting profiles are not valid.
- Do not define landing curves that overlap or are collinear.
- When placing profiles by projection, the default profile type depends on the selected sketching plane. This may not be the intended type. For example, when creating longitudinal profiles with a transverse sketch plane, the default type is **Transverse**. Check the type on the **Properties** dialog box, and modify it if needed.

- When placing profiles or seams by projection, extend the landing curves beyond the expected plate boundary or surface edges. If the geometry of the surface changes, then the landing curves are be long enough. The recommended extension is approximately 1 meter.
- When placing profiles or seams by offset, do not use an offset value of zero.
- When placing profiles or seams by projection that are to be bounded, extend the sketch curve past the boundaries in Sketch 2D and then set boundaries in 3D.

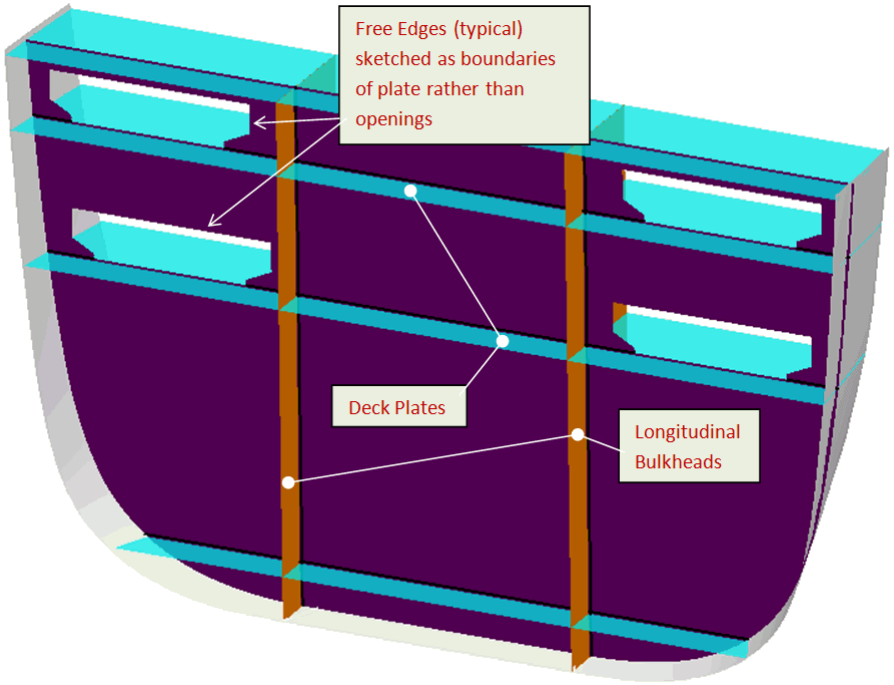
Also See

Placing Profile Systems (on page 721)
Creating Seams (on page 888)
Creating Reference Curves (on page 972)

Model Bulkheads with Multiple Openings

This best practice illustrates how to model interior openings as sketched boundaries and include the decks that define the base of the openings in the bulkhead boundary list, allowing the transverse bulkhead to be modeled as one plate system.

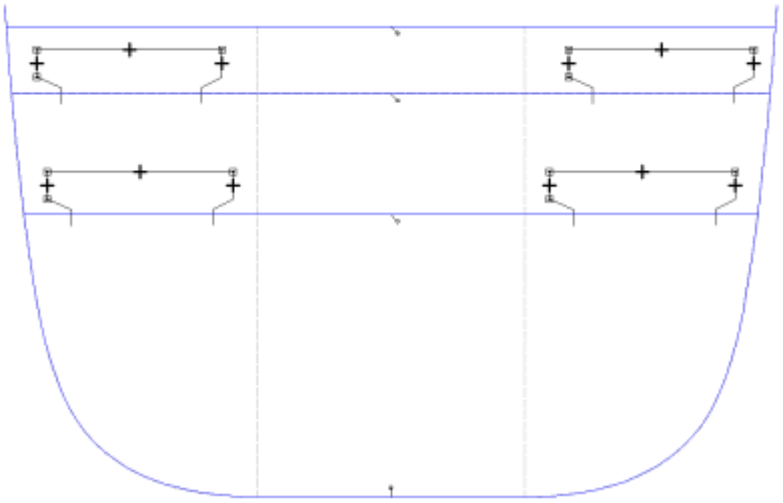
You want to create a transverse bulkhead as one plate system, which is split by the decks and longitudinal bulkheads, and contains large openings outboard of longitudinal bulkheads. The challenge in modeling the transverse bulkhead is how to model the openings. If you create an opening, the base of the opening is coincident with the intersection seam, which sometimes causes detailing to fail.



Instead of using **Place Opening** to model the openings, model the interior openings as sketched boundaries and include the decks that define the base of the openings in the bulkhead boundary list. The transverse bulkhead to be modeled as one plate system.

The following example uses sketched boundaries to create interior openings that are coincident with intersection seam:

1. Import the hull using *Imported Plate System* (on page 130) .
2. Assuming grid planes that are 1 meter apart, create decks D3, D21, D30, and D35, bounded by the hull and reference frames **x = 40** and **60M**, using *Planar Plate System* (on page 99) .
3. Create longitudinal bulkheads L10 and L-10, bounded by the hull and the D35 deck system, also using the **Planar Plate System** command.
4. Using **Planar Plate System** , create a transverse bulkhead at F50:
   a. Set **Continuity** to **Intercostal**.
   b. As the boundaries, select the hull, D21, D30, and D35, and then click **Sketch Boundary Curve** .
   c. Click **Add Intersecting Item** and select bulkheads L10 and L-10.
   d. Draw the opening boundaries as shown.

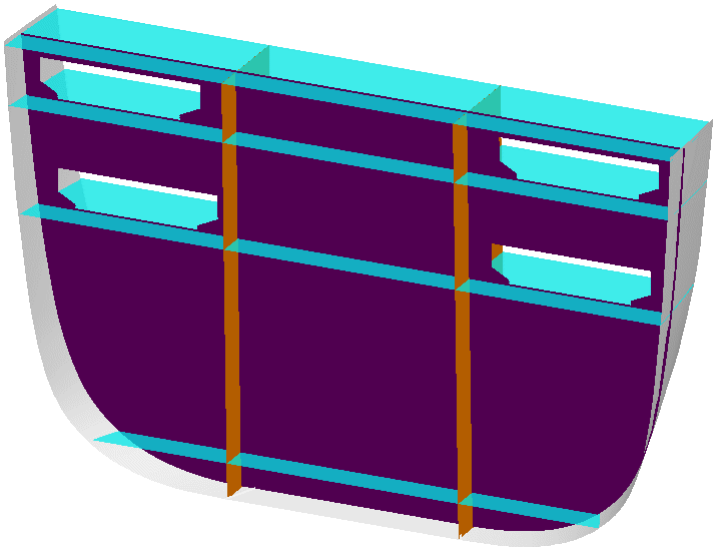


   e. Click **Finish**.

   The software starts the Ambiguity Solver. Three cells are presented (between hull and D21, between D21 and D30 and between D30 and D35).

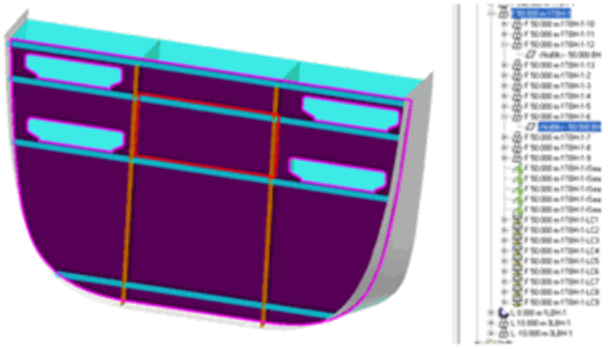
   f. Select all three cells, and then click **Finish** again.

The bulkhead is created as shown.

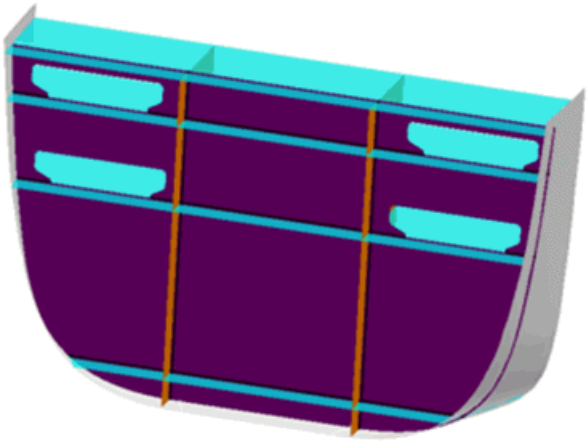


5. Run **Execute Split** .

   The software creates the 12 leaf systems and all the logical connections.



At this point you can detail the parts, create the manufacturing parts, add design seams to the bulkhead, and place edge reinforcement on the free edges of the sketched boundaries. You can also modify the sketched boundaries to have round corners:

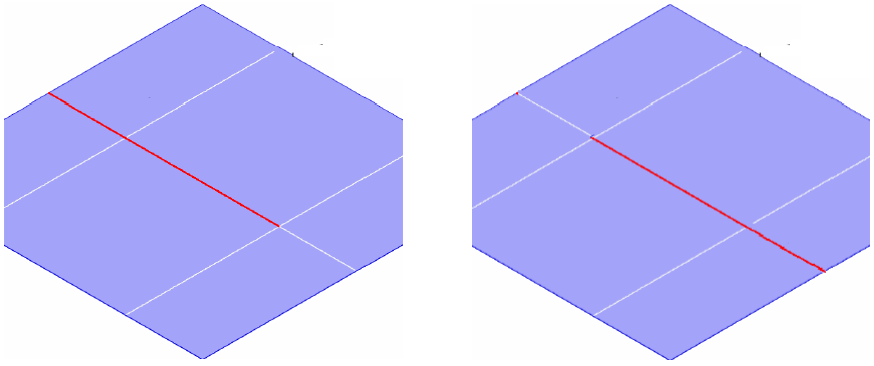


Seams Best Practices

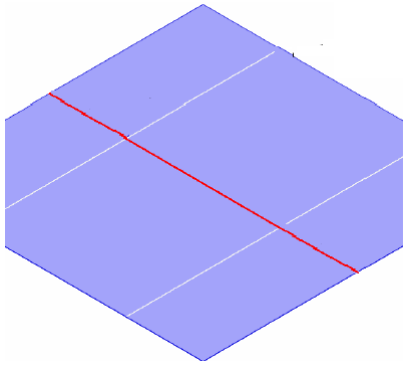
By using a seam object, you can split a plate system into separate child plate subsystems. A seam is always attached to one or more plate systems. Use the following best practices when working with seams. For more information, see *Creating Seams* (on page 888).

Avoid overlapping or coincident seams

Do not place seams that overlap. Overlapping seams can cause trimming issues. In the following example, two seams overlap.

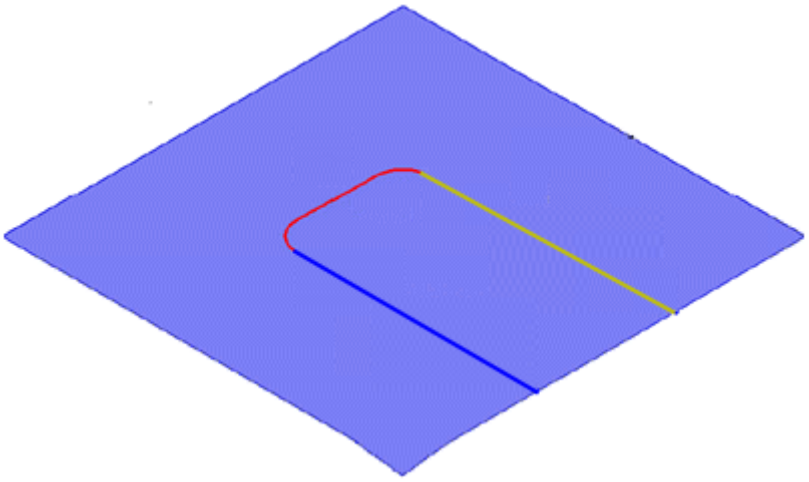


Instead, place a single seam.

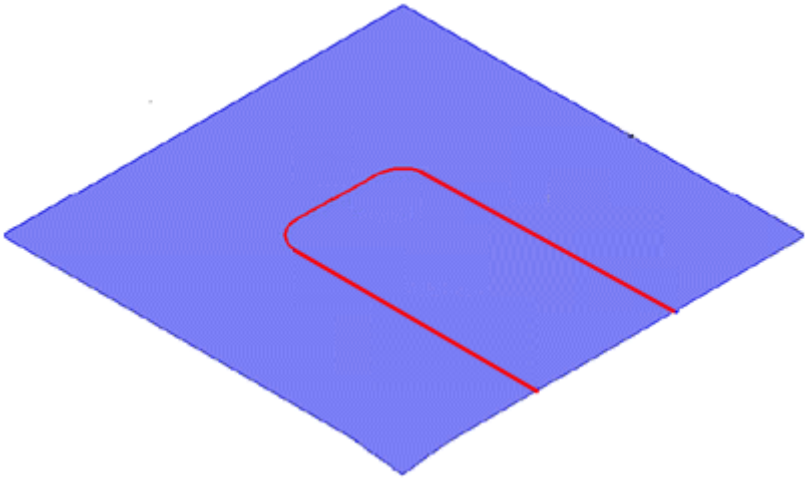


Avoid seam boundaries at tangent points

Seams at tangent points can cause problems when detailing. Do not place separate seams.



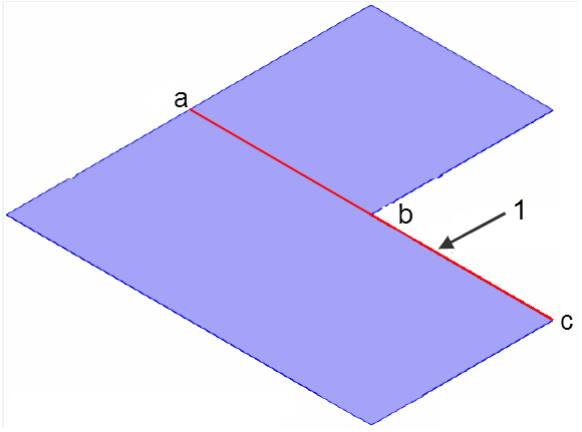
Instead, place a single seam to avoid seam boundaries at tangent points.



Avoid seams that are coincident with system boundaries

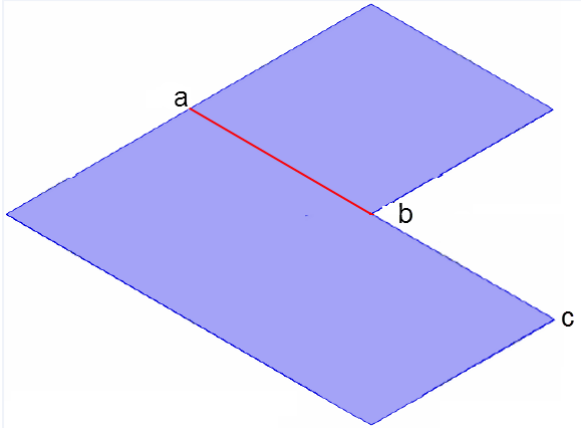
To avoid weld and other detailing issues, do not place seams that coincide with the system boundary.

In the following example, a seam from point **b** to point **c** is not required and makes detailing and trimming of the plate difficult.



1 - Seam coinciding with the plate boundary

Instead, place the seam only from point **a** to point **b**.



Avoid seams that end in the middle of the plate

To avoid detailing problems, do not place seams that do not completely split the plate. Place a seam across the plate touching the plate edges.

Seams by projection on non-planar plate systems

When you place a seam by projection on non-planar plate systems, extend the ends of the seam past the plate boundaries.

Seams by projection

When you place a seam by projection, connect the elements in Sketch 2D when the seam consists of multiple elements.

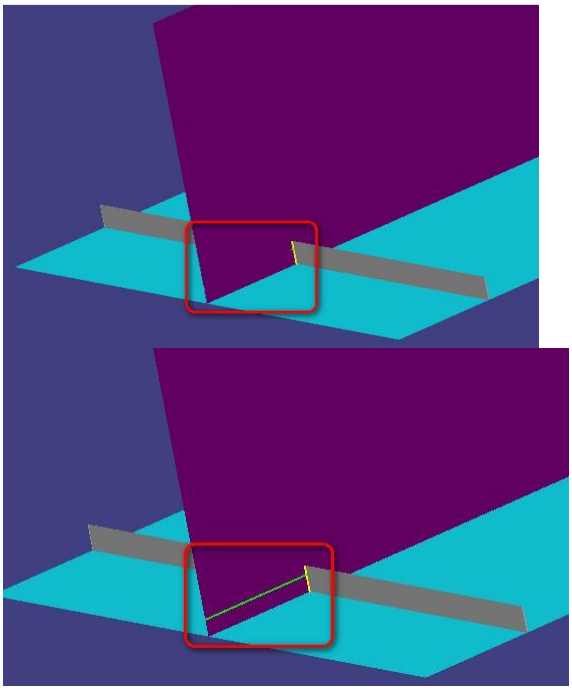
Split the intersection seam before creating a design seam

Split an intersection seam before creating a design seam that is bounded to the intersection seam.

In the following example, only a part of the plate is required to be cut. If the design seam is created without splitting the intersection seam, then the complete plate is split by the design

seam. If you split the profile seam first, then a design seam can be created that ends at the profile edge.

Split the profile to create the intersection seam, and then create the design seam and bound it to the intersection seam.



Closed edge reinforcements with seams must have at least two seams

You need to create at least two profile seams on a closed-edge reinforcement. For example, create one seam on the port side and the other on the starboard side. Select **Auto Split** and select the port solution to solve the ambiguity.

Continuous and intercostal seams

Create longer continuous seams before creating shorter intercostal seams. This helps in detailing and reduces ambiguity.

Modifying the seam geometry causes the seam to re-split

When you modify seam geometry, the software reruns the splitting operation on the seam. This might affect the objects that intersect with it. Plan seams carefully and proceed with caution if you need to modify a seam.

Session File Best Practices

A session file is a standard workspace file used to save settings from one work session to the next, such as settings for windows, default colors, Sketch 2D, and units of measure. Use the following best practices when working with session files. For more information, see *Managing Sessions* in the *Common User's Guide*.

Save Session Files

- After restoring new databases from a backup, open the model, make necessary setting changes, and save new session files.
- Save a session file after making significant changes, such as splitting blocks and detailing structure. This minimizes the refresh time the next time you open the model from the session file.
- You cannot save a session file if you run out of memory or the session quits abnormally. Re-open the session file and refresh the workspace to restore. If new objects are not defined in a filter, browse the database and add the objects to the filter.

Global Workshare

- When working in a global workshare environment, refresh the workspace after opening a session file.

Share session files

When sharing a session file, define the session file with catalog or model filters. You must have appropriate permissions to create or modify catalog filters and model filters. For more information, see *Permission Group Tab* and *Permissions Overview* in the *Common User's Guide*.

- Catalog filters are used like reference data in the catalog. A catalog filter applies to company-wide operations. For example, your administrator can define filters such as Company_Filter_1, Company_Filter_2.
- Model filters are available to everyone assigned to a specific database model.
- Delivered catalog filters query different types of model objects.
- You must have appropriate privileges to create, edit, or delete model and catalog filters.

★ **IMPORTANT** Do not create model filters at a satellite site.

For more information, see *Define Workspace* in the *Common User's Guide*.

Sketch 2D Best Practices

The **Sketch 2D** option is used in several commands to create sketched boundaries, openings, curves for extruded plate systems, and profile and seam landing curves. The following Sketch 2D best practices help you create objects in Molded Forms that you detail in the Structural Detailing task and manufacture in the Structural Manufacturing task.

Templates

Sketch 2D default display options are saved in the file *[Product Folder]*\Common2D\Symbol2D\Templates\template.sha. You can change the default display by changing options in this file:

1. Right-click template.sha and select **Properties**.
2. Clear **Read-only** and click **OK**.
3. Open template.sha and change any of the following default display options:
   - **File** > **Properties**. You can change the units displayed in command ribbon bars on the **Units** tab.
   - **View** > **Toolbars**. You can select the toolbars that are displayed.
   - **Format** > **Dimensions**. You can change the units used in dimensions, the text size, and other dimension properties.
   - **Tools** > **Options**. You can change the default colors on the **Colors** tab.
   - **Tools** > **Display Manager**. You can change the default geometry color, line type, and width for individual layers.
   - **Tools** > **SmartSketch Settings**. You can select the SmartSketch relationships that are enabled on the **Relationships** tab.
4. Save and close the file.

Fractional Units

The default units and precision values in Sketch 2D are defined in a 3D task. For example, if the file *[Product Folder]*\Common2D\Symbol2D\Templates\template.sha has default units set to **mm**, and the units of measure in Molded Forms are set to **ft-in**, Sketch 2D defines the units of measurement as **ft-in**.

The **Properties** dialog box in Sketch 2D allows you to view the defined units of measurement and to specify the fractional precision value. If you need to use a specific fractional precision value for all tasks in Sketch 2D, you can specify the precision value in the template file, which takes precedence over the precision value defined by the 3D task.

The following example shows how to modify the length readout precision to **1/64** in the template file:

1. In the template.sha file, set the precision value to **1/64**, and save the file.
2. In the Molded Forms task, click **Tools > Options**.

   *The **Options** dialog box displays.*
3. Under **Units of Measure**, set the distance readout to **ft-in (fractional)**.
4. Click **OK**.

*The distance units in Sketch 2D are now in **ft-in** with a precision value of **1/64**.*

Number of sketch files allowed per plate

Sketch 2D geometry for an object is saved in 2D files cached in the model and related to the object.

The **Boundary Curve** step for the plate system commands uses one 2D file. All boundary curves are in one 2D file for each plate system, and are modified, added, or deleted in the same file.

The **Place Opening**, **Stiffener Profile System by 2D Projection**, and **Design Seam by 2D Sketch** commands can each have:

- Multiple 2D files, with a separate file for each object on one plate system. The objects are modified in the 2D file, and deleted from the graphic view or **Workspace Explorer**.
- Geometry for multiple objects in one 2D file. The objects are modified or deleted in the 2D file.
- One 2D file that applies to multiple plate systems. Changes to the 2D file are reflected on all plate systems.
- A combination of these files.

Add items to sketch

The **Add Intersecting Item** and **Add Projection Item** steps are used to add reference structure to Sketch 2D. Select the minimum number of needed items.

NOTE More than 30 items added to a sketch file can slow down software performance when the object is recomputed.

Next, you can select **Auto** to automatically add all objects that are relative to the object to be sketched to the current select set. Items that pass the criteria are added to the select set and are highlighted in the graphic view, as well as the **Workspace Explorer**.

The filter criteria that the software uses to determine eligible candidates depend upon whether **Add Intersecting Item** or **Add Projection Item** was selected in a previous step. The following table lists the various relevant objects that are automatically imported based on the target, or base sketch, object type.

Base Sketch Object	Intersecting Objects	Projecting Objects
Plate System	▪ Stiffeners ▪ Edge Reinforcements ▪ Seams ▪ Bounded Objects ▪ Bounding Objects ▪ Knuckle Reference Curves	▪ Reference Curves ▪ Stiffeners ▪ Penetration Plates
Plate Part	None supported in the current version of the software	None supported in the current version of the software
Profile System	None supported in the current version of the software	None supported in the current version of the software
Member	None supported in the current version of the software	None supported in the current version of the software

Designed Member (Built Up)	Same as Plate System	Same as Plate System

If no objects are added to the select set, the software displays a message on the status bar.

If you clear **Auto,** all selected objects, whether added to the select set automatically or manually, are cleared. Selecting **Auto** adds back those objects that initially passed the filter criteria, but it does not add back any objects that you selected manually.

⚠ CAUTION When the selected plate system is large, **Auto** automatically adds all objects relative to the sketch in the context of *intersecting* or *project*. This can slow down software performance.



Layers

The software automatically defines several layers in Sketch 2D. Reference items (selected automatically by the software or manually using the **Add Intersecting Item** and **Add Projection Item** steps) are placed on those layers:

Default Layer

The software only uses geometry that you create on the **Default** layer. You can put other geometry, such as construction lines, on other layers. The Default layer must be the first layer alphabetically. To avoid a conflict, you should add a "z" prefix to any layer which you add, such as "z_construction."

Infinite Elements Layer

Contains reference items with infinite geometry intersecting the sketch plane, such as grid planes. Objects on this layer can be used with dimensions and relationships.

Inputs Layer

Contains other reference items intersecting the sketch plane that can also be used with dimensions and relationships. The reference geometry includes openings, design seams, plate systems, and profile systems.

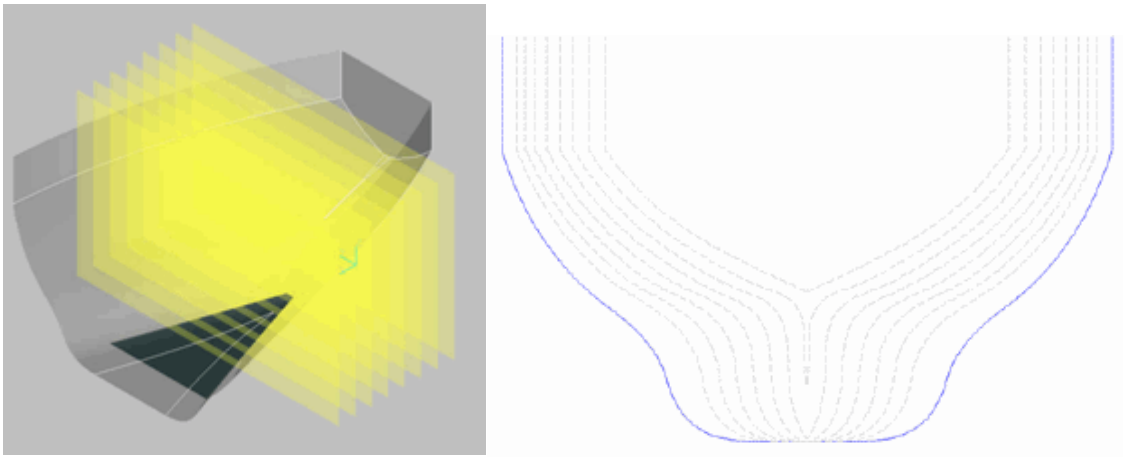
No_Constrained_Elements Layer

Contains reference items that do not intersect the sketch plane. Also contains reference items that cannot be used with relationships and dimensions, such as plate and profile parts. By default, you cannot select reference geometry on the **No_Constrained_Elements** layer. To make the geometry selectable, you must make **No_Constrained_Elements** the active layer, and then switch back to the **Default** layer.

You can add relationships and dimensions to reference geometry on the **No_Constrained_Elements** layer, but the relationships are not saved when you click **Finish**.

Multiple reference planes

You can add reference planes to Sketch 2D that are different than the sketching plane, but parallel to the sketching plane. This is useful in defining landing curves for projected profiles and seams on the hull. Use **Add Projection Item** to add the reference planes.



The additional reference plane intersection geometry is gray in color and placed on the **No_Constrained_Elements** layer.

Using SmartSketch Drawing Editor to save sketch geometry

SmartSketch Drawing Editor, delivered to *[Product Folder]*\Shape2D\Bin\shape2dserver.exe, can be used to save geometry commonly used in Sketch 2D, such as:

- Mother curves and extrusion curves for extruded plate systems.
- Boundary curves.
- Openings or sketched features not included in the Catalog.
- Landing curves for projected profiles and seams.

You can copy the geometry from Sketch 2D and paste it into SmartSketch Drawing Editor, or create the geometry directly in SmartSketch Drawing Editor. Geometry can be saved in multiple SmartSketch Drawing Editor files or into a single file with multiple sheets. The geometry can be copied from SmartSketch Drawing Editor and pasted into Sketch 2D as needed.

Deleting reference geometry

You can delete reference geometry from the sketch when the software includes geometry that you do not want to see. The reference structure is not deleted, and the reference geometry is recreated the next time that you open the sketch.

Modifying groups of geometry

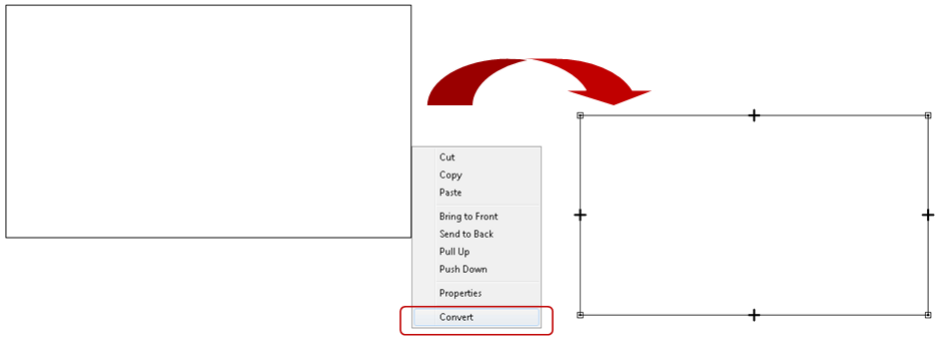
The software automatically groups Sketch 2D geometry when you click **Finish**. When you modify the existing 2D geometry of an object, you must maintain the original group identifier by using the following methods:

1. To modify or delete existing geometry, use **QuickPick**, **Top Down** or **Bottom Up** with **Select Tool** to select the geometry instead of the group.
2. To add new geometry to a group, use **Edit** > **Group Modify**.
3. To replace all geometry in a group with geometry copied to the clipboard, use **Replace Group Elements**.

Rectangles

Use **Line** ⟋ or **Line/Arc Continuous** to create rectangles.

If you use **Rectangle** ▭, you must select **Tools** > **Maintain Relationships**. Right-click the geometry, and then select **Convert**. This converts the rectangle into four lines and the correct connection relationships are created at the corners.



Constraints

Use relationships and dimensions to constrain geometry to reference structure. This keeps the geometry properly related to the reference structure if the reference structure changes.

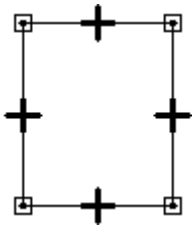
Keep relationships and dimensions as simple as possible. This allows the geometry the best chance to update if the reference structure changes.

Connections

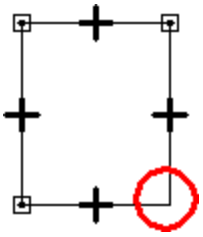
Connection relationships are required for the software to create connected geometry.

Do not create a connection between the corner of the sketched geometry, and the corner of the bounding object. Instead, move the constraint slightly off the corner.

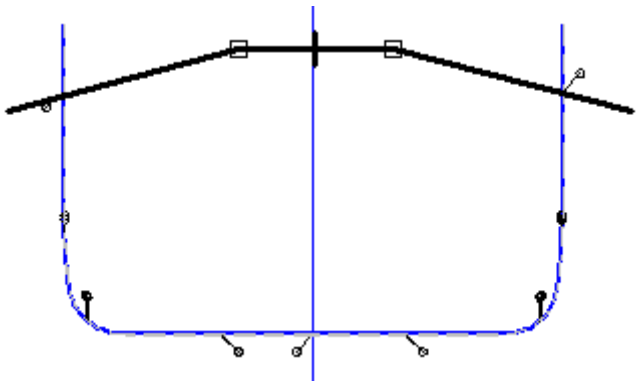
Good: All endpoints of opening are connected.



Bad: Missing connection at one corner causes the creation of the opening to fail.



Good: All endpoints of extrusion curve are connected.



Bad: Missing connection on extrusion curve.

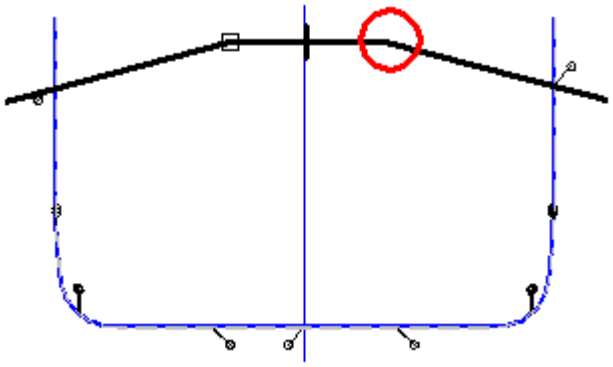
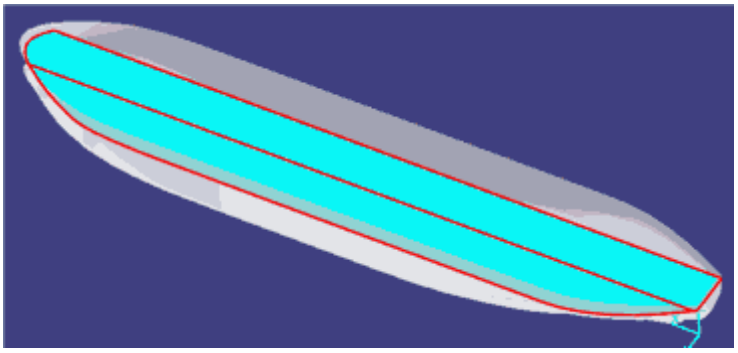


Plate system is missing part of the curve.

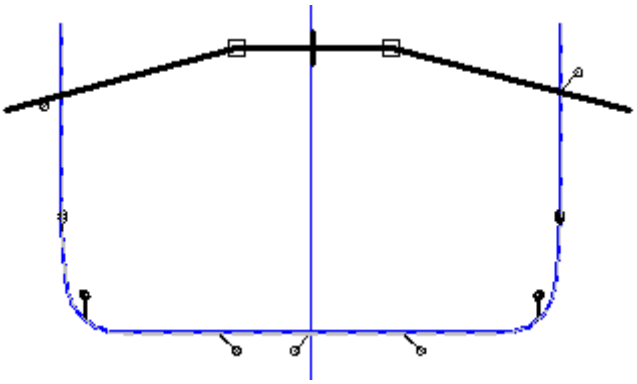


Extending sketch geometry to intersect bounding structure

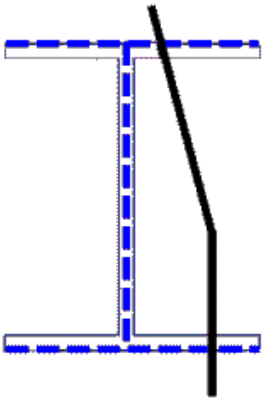
Extend sketch geometry beyond bounding reference structure geometry to ensure that the sketch geometry intersects the reference structure at all locations.

Good:

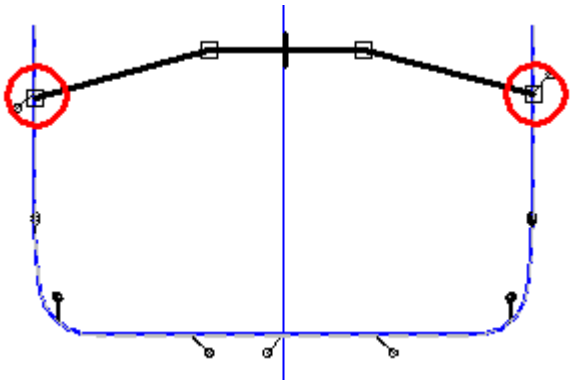
Extrusion curve extends beyond the hull.

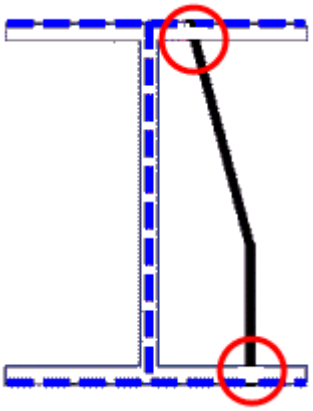


Bracket boundary curve extends beyond the top flange and the bottom of the bottom flange.



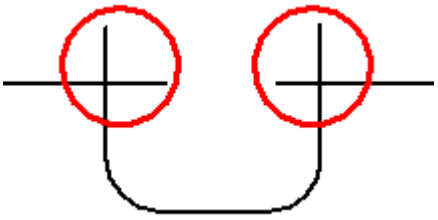
Bad:





Overlapping sketch geometry for boundary curves

A boundary curve with all geometry connected is treated as one edge in the Graphics View. You may sometimes want parts of the curve to be treated as separate edges at sharp discontinuities such as corners. This allows you to add edge reinforcements in the Molded Forms task or edge treatments in the Structural Detailing task to part of the boundary curve without defining additional boundaries. In order to separate the boundary curve into separate edges, overlap the boundary curve geometry instead of connecting it at the corners:

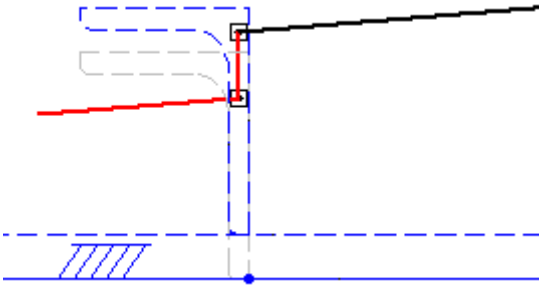


Creating boundary curves on the thickness side of a plate

If your plate boundaries include a plate system with the thickness direction towards the plate that you are creating, a profile mounted to the bounding plate system and a sketched boundary, and then the boundary curve must intersect both the profile system and profile part.

In the following example:

- The solid blue line is the molded surface of the bounding plate system.
- The dashed blue line is the thickness direction for the bounding plate part.
- The dashed gray profile outline is the bounding profile system mounted to the bounding plate system at the molded surface (on the **No_Constrained_Elements** layer).
- The dashed blue profile outline is the bounding profile part mounted to the thickened bounding plate part.
- The solid black line is the intended boundary curve intersecting the profile part.
- The solid red line is the required adjustment to the boundary curve, passing through both the profile part and profile system, and intersecting the profile system.

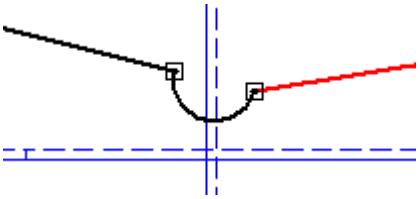


The black line is the intended boundary curve. It can be constrained to the dashed blue profile part using relationships and dimensions. The red lines are additional geometry that must connect to the black boundary curve, pass through the profile part geometry, and intersect the dashed grey profile system geometry.

Arcs in boundary curves

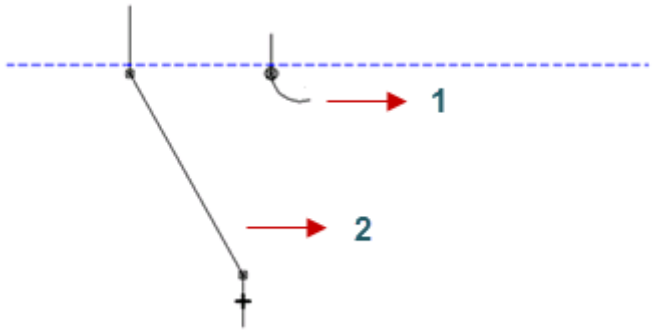
If a boundary curve ends with a circular or elliptical arc, you must add a line segment pointing away from the direction of the arc. This prevents the arc from extending back onto the boundary curve as the curve is calculated.

In the following example, the black curves are the intended boundary curve. The red line is the extension to the boundary curve that prevents the arc from extending back onto the boundary curve.



Multiple curves

Do not create a close boundary curve contour that overlaps trimming input geometry. Instead, draw two separate curves. In the following example, there are two separate curves, one for corner feature and one for diagonal boundary.



1 - Corner feature curve

2 - Diagonal boundary curve

Associative offsets

The **Associative Offset** command allows you to create a curve based on an existing curve, and place the new curve at a specified offset. To use this command, you must add it to a toolbar.

1. Right-click an existing toolbar, and click **Toolbars**.
2. Click **Customize** on the **Toolbars** dialog box.
3. Click **Manipulation** on the **Categories** list.
4. Drag the **Associative Offset** icon from the **Buttons** list to an existing toolbar.

For more information on the **Associative Offset** command see the **Associative Offset Command** topics in the *SmartSketch Drawing Editor* help file.

Split Best Practices

Execute Split divides large plate and profile systems into smaller subsystems. The systems can be split by intersection and design seams. **Execute Split** automatically creates intersection seam at the intersection of two objects. Use the following best practices when working with the command. For more information, see *Execute Split* (on page 891).

Continuities and priorities

After you place a plate or a profile, specify continuity and priority before splitting.

1. Select plate or profile system. You can select more than one system if they are to have the same continuity and priority.
2. Click **Properties** on the ribbon bar.
3. On the **Main** tab, specify **Continuity** and **Priority**.
4. Click **OK**.

Continuity defines how the object reacts when it intersects another object. Select **Continuous** to indicate that the object penetrates the other object. Select **Intercostal** to indicate that the object is penetrated by the other object.

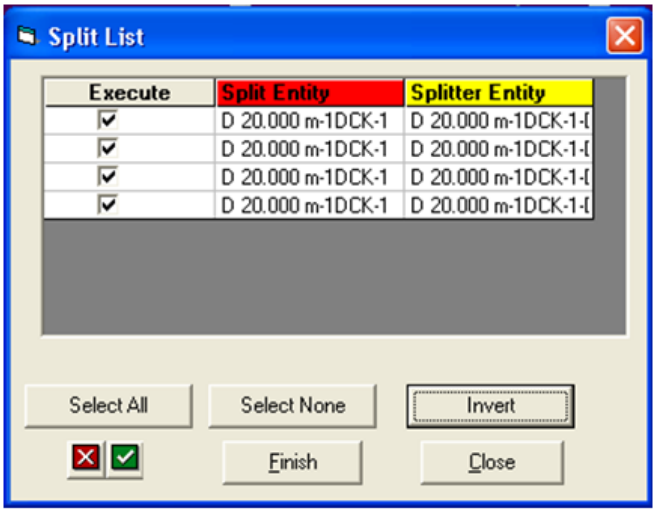
Priority specifies which object is penetrated when two similar objects with the same continuity value intersect. An object with a lower continuity priority (for example, 1, 2, or 3) penetrates other object with a higher continuity priority (for example, 7, 8, or 9).

Design seams

Split intercostal structure with continuous structure before placing design seams on the intercostal structure. For best performance, create all design seams, and then execute split before detailing for best performance.

Review the Split List

Each object by default is set as a split (intercostal) or splitter (continuous) object based on the continuity and priority properties you specify.



1. If a row in the split list is highlighted in red, there is ambiguity in the split-splitter relationship. Modify the continuity and priority properties of the objects as needed.
2. Click **Invert** to swap split and splitter objects.
3. Click **Finish** after reviewing the list.

NOTE Split the objects in batches rather splitting entire objects at the same time. Splitting early and often reduces the computation time.

Splitting late in design

When modifying or creating a split after detailing, many parts are affected. In this case, use the following workflow:

1. Turn on **Delay Struct Detailing**
   a. Go to **Tools** > **Delay Settings**.
   b. On the **Delay Setting** tab**,** select **Delay Struct Detailing**.
   c. Click **Apply**, and close the dialog box.
2. Split
   a. Select the objects to split or a split to modify.
   b. Click **Execute Split**  .
   c. Review the split list as needed, and then click **Finish**.
3. Turn off **Delay Struct Detailing**
   a. Go to **Tools** > **Delay Settings**.
   b. On the **Delay Setting** tab**,** clear **Delay Struct Detailing**.
   c. Click **Apply**.
4. Update
   a. On the **Update To Do Records** tab, click **Update Objects**.
   b. Click **OK** to close the dialog box.

Un-Split

Use **Un-split** command to remove a split. You can use the seam that caused the split again at a later time.

1. Select **Tools** > **Un-Split** from the menu.
2. Click **Select**  on the ribbon.
3. Select the seam that caused the split to undo.
4. Click **Un-Split**.

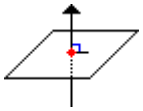
Plane Methods

You must often define planes used for object surfaces and sketching planes. Usually, but not necessarily, these planes are based from an existing plate or from a grid or reference plane created using the Grids task. The following methods define planes:

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

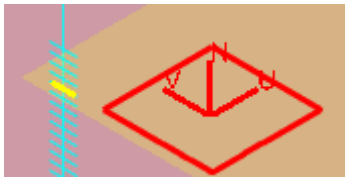
What do you want to do?

- *Define a coincident plane* (on page 76)
- *Define an offset plane* (on page 77)
- *Define a plane using a point and a normal vector* (on page 78)
- *Define a plane using three points* (on page 79)
- *Move a defined plane* (on page 80)

Define a coincident plane

1. Click **Offset from a Plane** .
2. In the model or **Workspace Explorer**, select a reference plane, a grid plane, or a planar plate system.

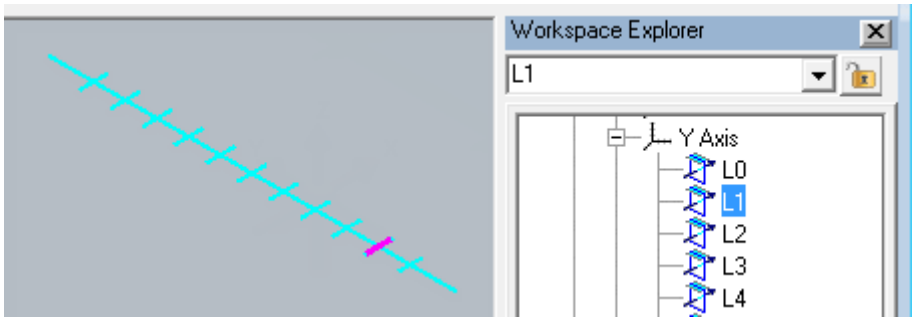
   A preview of the selected plane displays. The local coordinate system of the plane displays at the plane's center.



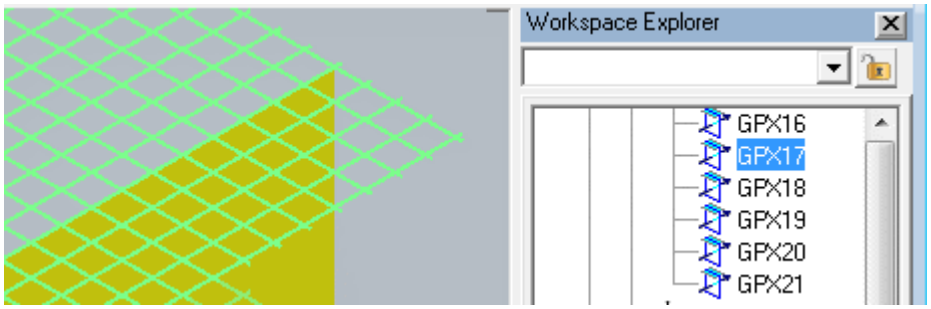
3. Check that **Offset Lock** is locked and that the **Offset** value is 0.

NOTES

- You create reference planes in the Grids task by selecting **Ship** in the **Grid Type** box of the **Grid Wizard**. For this plane method, you can select reference planes in the **Workspace Explorer** or in a graphic view.



- You create grid planes in the Grids task by selecting **Grids** in the **Grid Type** box of the **Grid Wizard**. For this plane method, you can best select grid planes in the **Workspace Explorer**.

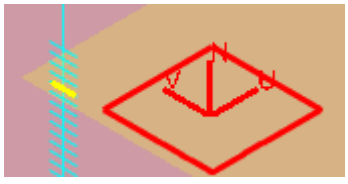


- For more information, see *Create Coordinate System (Grid Wizard)* in the *Grids User's Guide*.

Define an offset plane

1. Click **Offset from a Plane**.
2. In the model or **Workspace Explorer**, select the base plane from which to offset the new plane. The base plane can be a reference plane, a grid plane, or a planar plate system.

   A preview of the selected plane displays. The local coordinate system of the plane displays at the plane's center.



3. In the **Offset** box, type the offset distance from the plane and press TAB.

   The plane moves to the new location.

   -OR-

4. Graphically define the offset:

a. Click **Offset Lock** .

The option changes to unlocked .

b. Move the pointer to the required location and click.

*The plane and plane coordinate system move to the new location, **Offset Lock** changes to locked , and the offset value displays in the **Offset** box.*

c. If an adjustment to the offset is needed, type a new value in the **Offset** box.

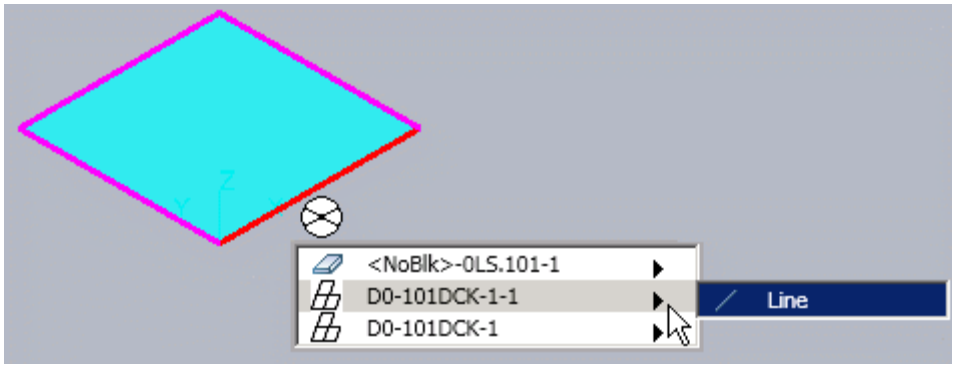
5. To move the plane independent of its previous definition, see *Move a defined plane* (on page 80).

TIPS

- You can graphically define the offset with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.
- By moving the pointer back and forth over the base plane in the graphic view, you can change the offset direction. You can also change the direction by changing the sign in the **Offset** box.
- When **Offset Lock** is set to locked , the **Offset** value does not change when the pointer moves.

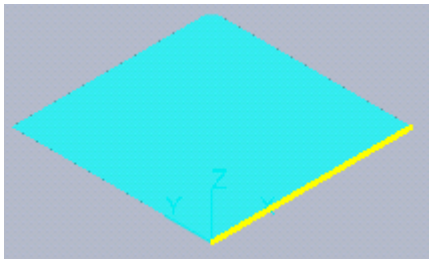
Define a plane using a point and a normal vector

1. Click **Plane by Point and Vector** .
2. Select a vector in the model that is normal to the required plane. The vector can be any linear geometry that is part of the construction of a model object, such as a plate system edge or a profile system landing curve. Use **QuickPick** to select a line geometry element.



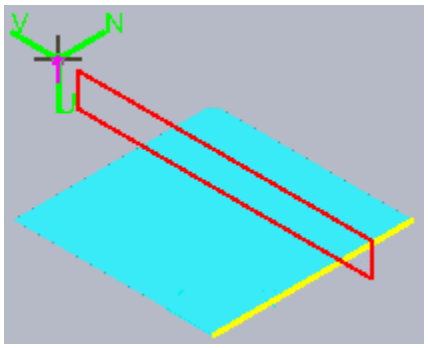
3. Click the **Line** geometry element.

The selected geometry highlights in yellow.



4. Move the cursor to a point that is on the required plane, and then click.

   A preview of the plane and its local U_V_N coordinate system displays. The plane is coincident with the point and normal to the vector.



5. To move the plane independent of its previous definition, see *Move a defined plane* (on page 80).

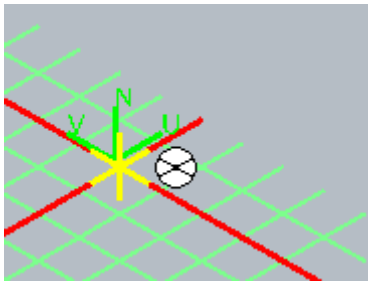
TIP You can graphically define the point with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.

Define a plane using three points

1. Click **Plane by Three Points**.
2. Specify on the first point that defines the plane.
3. Specify the second point that defines the plane.
4. Specify the third point that defines the plane.

NOTES

- You can define points on the surfaces, edges, and corners of systems and parts. You can control the types of points by using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.
- You can define points on grid planes, especially at intersections.



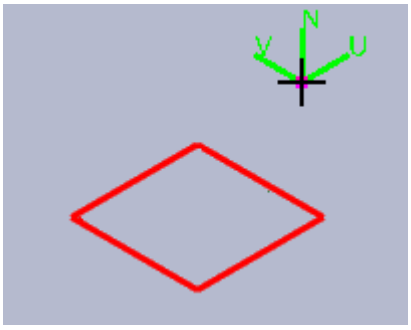
You can control the types of points using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

In Molded Forms, define parametric points by using **Insert** > **Topological Points**. For more information, see *Topological Points (Insert Menu)* (on page 1323).

Move a defined plane

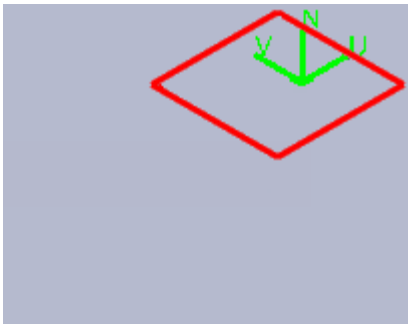
1. Click **Maintain Plane Relationships** to disable relationships used to create the plane.
2. Click **Move Plane**.

   The pointer changes to a plus symbol, and the origin point of the local U-V-N coordinate system for the plane moves to the pointer.



3. Move the pointer to the required location, and then click.

   The local U-V-N coordinate system and the plane display in the new location.



TIP You can move the plane with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.

Boundary Methods

You must often define boundaries when creating plate systems, profiles, edge reinforcements, and other objects. The common methods for defining boundaries are:

Pick Boundaries

Define boundaries by selecting them graphically either in a graphics view or in the **Workspace Explorer**.

Pick Edges

This option is only available if you pick a plate as a boundary. Select the plate edges to use. Click **Pick Boundaries** to exit the **Pick Edges** option and select other boundaries.

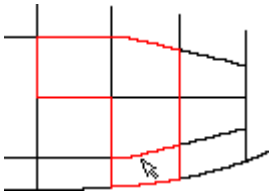
Boundary List

Define boundaries by selecting them from a list of potential boundaries. For more information,

see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Define the boundary by drawing its shape.



Solve Ambiguity

If the boundaries that you selected can form the object in more than one way, then you have defined an ambiguous solution. The software prompts you to select one or more solutions.

What do you want to do?

- *Pick boundaries* (on page 81)
- *Use the Boundary List dialog box* (on page 81)
- *Define a boundary curve* (on page 82)
- *Solve ambiguous solution created by selected boundaries* (on page 82)
- *Rebound objects* (on page 1391)

Pick boundaries

1. Click **Pick Boundaries** .
2. In a graphic view or **Workspace Explorer**, select an object or grid plane to use as a boundary.
3. Continue selecting objects or grid planes until you have completed defining the boundaries for the object.

NOTE To remove an object from the group of boundaries, select the object again.

Use the Boundary List dialog box

1. On the command ribbon, click **Boundary List** .

   *The **Boundary List** dialog box displays.*
2. To remove a boundary, select the boundary in the list, and then click **Remove** .
3. To remove all boundaries, click **Remove All** .
4. To add a new boundary, click **Add** , and then select a boundary in the graphic view or in the **Workspace Explorer**. You can also type the boundary name and an optional offset value in the **Offset** box or the **Name** box.

NOTE For more information, see *Boundary List Dialog Box* (on page 83).

Define a boundary curve

1. Select **Boundary Curve** .
2. Define the sketching plane for the curve.

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define plane using angle from plane
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)
   Define plane by vectors normal
3. Click **Add Intersecting Item** .
4. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the 2D environment.
5. Click **Add Projection Item** .
6. Select objects in the model that do not intersect the sketching plane and that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the 2D environment.
7. Select **Auto** to automatically add all related objects to the select set, if needed.
8. Click **Sketch 2D** .

   The 2D environment appears.
9. In the 2D environment, sketch the boundary.

   NOTE You can constrain the boundary to geometric construction points referenced from profile cross-section key points or offset from plate systems. For more information, see *Point at Offset from Keypoint Command* and *Point at Minimum Distance Command* in the *SmartSketch Drawing Editor Drawing Editor Help*, available within the 2D environment.
10. Click **Finish** in the 2D environment.

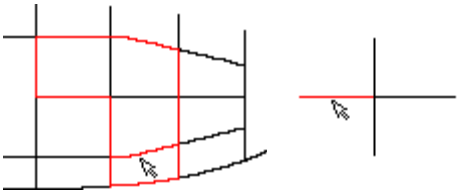
    The 3D environment and the main command ribbon return.

NOTE Deleting a boundary curve from the 2D environment also removes it from the **Boundary List** dialog box.

Solve ambiguous solution created by selected boundaries

The command switches to **Solve Ambiguity** automatically if an ambiguous solution exists after clicking **Accept** or **Finish**. All possible solutions created by the selected boundaries appear in the graphic view.

1. Move the pointer over a solution, and then click to select it.
2. Continue to select solutions until you have defined the entire bounded object. A correct solution to ambiguity meets these conditions:
   - Each solution must have a common edge with at least one other solution.
   - Each boundary must be used by at least one solution.



NOTES

- To modify the ambiguous solution after creating an object, select **Solve Ambiguity** , and select new solutions.
- To remove a previously-selected solution, select the solution again.
- Pressing the SHIFT key selects the **Pick Edges** option after you have selected a boundary plate. Press SHIFT again to return to the **Pick Boundaries** option.

Boundary List Dialog Box

Displays the boundaries for the object that you are placing:

- Model objects, such as plate systems or profile systems.
- Reference planes and other plane definitions.
- Sketched boundary curves created using the **Sketch 2D** option on a supported ribbon. Boundary curves are prefixed with **SketchGroup_.**
- A standalone plane created during a copy, move, or mirror operation when a boundary was not included in the operation. You usually want to replace the standalone plane with a new boundary.

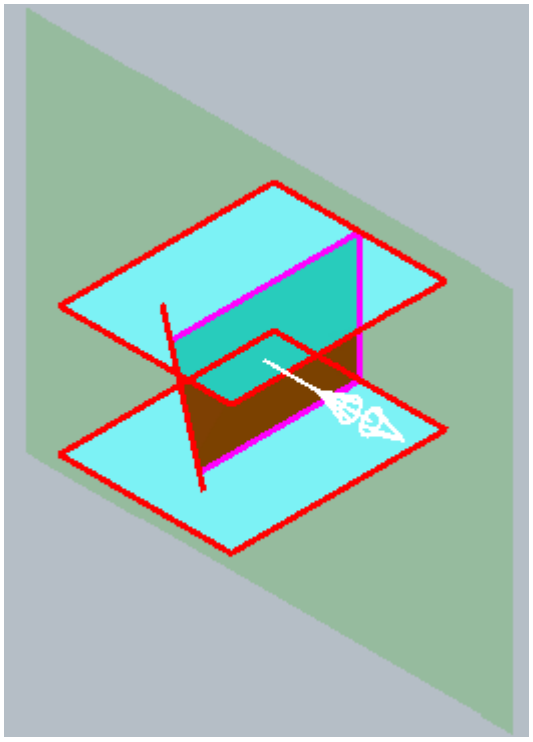
Open this dialog box by clicking **Boundary List** on the command ribbon during the boundary definition step.

Preview Type

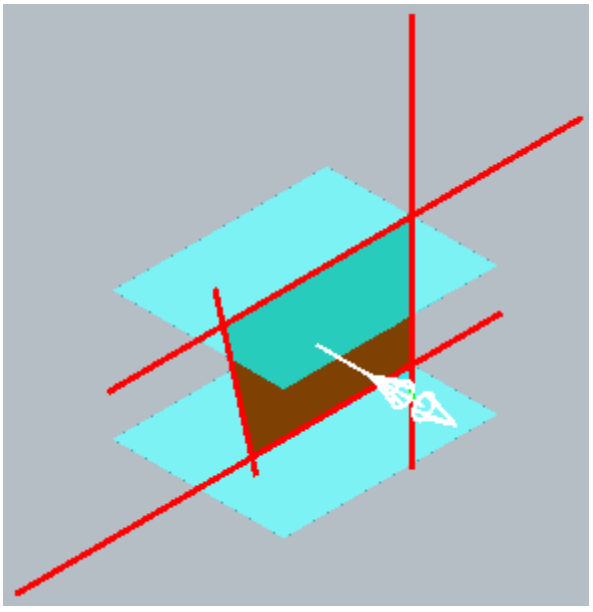
You can press ARROW DOWN ↓ and ARROW UP ↑ to step through the boundary list. Each selected boundary highlights in the graphic view according to the selected preview type. **Tools** > **Options** defines the **Highlight** color. For more information, see *Colors Tab (Options Dialog Box)* in the *Common User's Guide*.

In the following examples, two plate systems, a reference plane, and a sketched boundary are the selected boundaries of the vertical plate.

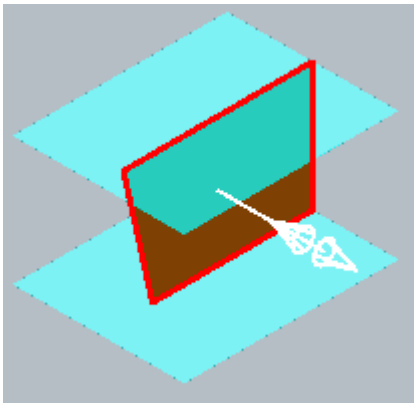
Bounding Objects - Highlights the selected boundary object as an outline. Reference planes display as translucent shaded planes.



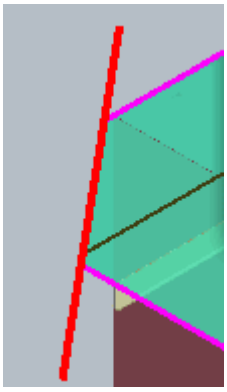
Bounding Geometry - Highlights the internal boundary geometry used by the software. The internal geometry is typically an unlimited length line for model objects and reference plane boundaries.



Edges - Highlights edges of the bounded object where the bounded object intersects with the selected boundary.



NOTE For **Bounding Objects** and **Bounding Geometry**, the complete sketched boundary curve geometry displays as drawn in **Sketch 2D**, as shown in the following example.



Add

Specifies a new boundary. You can select a boundary in the graphic view or the **Workspace Explorer**.

NOTES

- You can also type the boundary name and an optional offset value in the **Name** box or the **Offset** box, and then press TAB or ENTER.
- Invalid text displays with a gray background in the **Name** box or the **Offset** box.

Remove

Removes the selected boundary from the list.

NOTE Deleting a boundary curve from the 2D environment (**Sketch 2D** on the main command ribbon) also removes it from the **Boundary List** dialog box.

Clear All

Removes all boundaries from the list.

Disconnect

Removes the relationships of a boundary to its definition method. When you disconnect a boundary, it becomes a standalone plane. This option is active by default for a standalone plane created during a copy, move, or mirror operation.

NOTE After disconnecting a boundary, you must select a replacement boundary in the graphic view or the **Workspace Explorer**.

Extend Sketch Boundary in Graphic View

Extends a sketched boundary to a point in space or to another element selected in the graphic view. This option is available when you select a sketched boundary (with a name prefixed with **SketchGroup**_).

Name

Specifies the name of a boundary.

Offset

Specifies an offset distance from the boundary. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

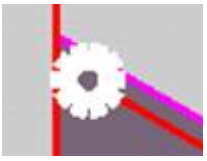
  Example (when the model has one coordinate system): **F20 + 1.5 m**

NOTE Invalid text displays with a gray background in the **Name** box or the **Offset** box.

Apply

Displays a preview of the boundaries in the graphic view.

NOTE Gaps between boundaries display with a glyph.



The glyph uses the **Handle** color defined in **Tools** > **Options**. For more information, see *Colors Tab (Options Dialog Box)* in the *Common User's Guide*.

OK

Closes the dialog box and saves the boundary changes.

NOTE To cancel boundary changes, you must click **OK** and then exit the parent command without clicking **Finish**.

Landing Curve Methods

You must often create landing curves when creating profile systems and seams by projection. Landing curves are sometimes based on existing grid planes created with the Grids task. There are two methods for defining the landing curve:

Define using 2D Sketcher

Defines a landing curve by projecting a 2D curve that is sketched onto the plate systems.

Landing Curve Definition by 3D

Defines a 3D geometric construction landing curve by creating a curve through a set of points, a set of curves on the plate system, or a surface intersecting the plate system. For more information, see *Geometric Constructions* (on page 1027).

Landing Curve Best Practices **(on page 56)**

What do you want to do?

- *Define landing curve by 2D* (on page 87)
- *Define landing curve by 3D* (on page 88)

Define landing curve by 2D

1. Click **Define using 2D Sketcher** .
2. Define a sketching plane using one of the following methods:

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)
3. Click **Add Intersecting Item** .
4. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the 2D environment.
5. Click **Add Projection Item** .
6. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the 2D environment.
7. Click **Sketch 2D** .
8. Sketch the 2D stiffener profile landing curve that you want the software to project onto the plate system that you are stiffening. For more information, see *Sketch 2D Best Practices* (on page 65).

   NOTE You can constrain the landing curve to geometric construction points offset from plate systems. For more information, see *Point at Minimum Distance Command* in the *SmartSketch Drawing Editor Drawing Editor Help*, available within the 2D environment.
9. Click **Finish**.

Define landing curve by 3D

NOTE You can create the landing curve by using a variety of macros available in the **Geometric Construction Palette** dialog box. The following procedure provides an example.

1. On the **Stiffener Profile System by Projection** ribbon, click **Landing Curve Definition by 3D** .

   *The **Geometric Construction Palette** dialog box, **Geometric Construction Explorer** dialog box, and **Geometric Construction** ribbon display.*

2. In the **Points** tab of the **Geometric Construction Palette** dialog box, select a point macro, and create the needed geometric construction points.
3. Click **Continue**.

   *The **Geometric Construction Explorer** shows a list of points.*

4. In the **Curves** tab, select the **CurveByPointsOnSurface** macro, and create the curve.
5. Click **Continue.**

   *The curve is added to the **Geometric Construction Explorer**.*

6. Click **Close**.

Profile Orientation

You must often define the orientation of stiffener profiles. A profile has a linear or twisted orientation and can be divided into multiple regions, each having a different orientation. The following options are available to define profile orientation:

Rule Region Definition

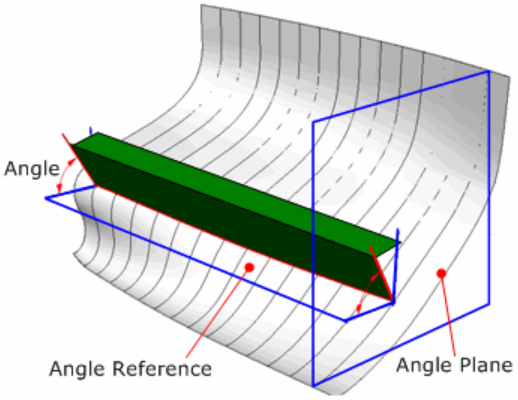
Defines regions along the profile landing curve. Each region of the landing curve can have a different orientation rule assigned to it. The software initially defines a region boundary point at the profile boundaries. You cannot delete or modify the software-defined region boundary points. You can define and modify additional region boundary along the profile landing curve using grid/reference planes, plate design seams, and plate planning seams.

Orientation Rule Assignment

Opens a list of orientation rules from which you can select. These rules include:

- **Normal** - The web of the profile follows the plate system surface normal along the landing curve. This is the default setting if the profile parent is a plate system.
- **Up-Down** - The orientation vector of the profile cross-section remains in an up-down direction. For example, you can select **Up-Down** for profiles on a deck.
- **Fore-Aft** - The orientation vector of the profile cross-section remains in a fore-aft direction. For example, you can select **Fore-Aft** for profiles on a transverse frame.
- **Port-Starboard** - The orientation vector of the profile cross- section remains in a port-starboard direction. For example, you can select **Port-Starboard** for longitudinal stringer profiles.

- **Planar** - The orientation angle remains constant relative to a reference plane that you specify.



- **Angle** - Type the angle for the plane.
- **Reference** - Select the reference plane from which to measure the angle. You can select **Normal**, **Up-Down** (horizontal), **Fore-Aft** (longitudinal) or **Port-Starboard** (transverse). **Normal** reference plane is the plate system surface normal along the landing curve and is available for any profile orientation. **Up-Down**, **Fore-Aft** or **Port-Starboard** are available as is appropriate for the direction of the landing curve.
- **Plane** - Select the plane *in* which to measure the angle. For **Normal** Reference, this option is unavailable because the angle is always measured in a plane normal to the landing curve. For **Up-Down** or **Port-Starboard** Reference, you can select **Normal** or **Orthogonal**. If **Normal** is selected, the angle is measured in a plane normal to the landing curve. Orthogonal is the transverse, longitudinal or horizontal plane closest to the landing curve normal. If **Orthogonal** is selected, the software selects the orthogonal plane based on average direction of the landing curve.

Twist Linear Girth

Twist is distributed evenly along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate is constant.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Twist Cubic

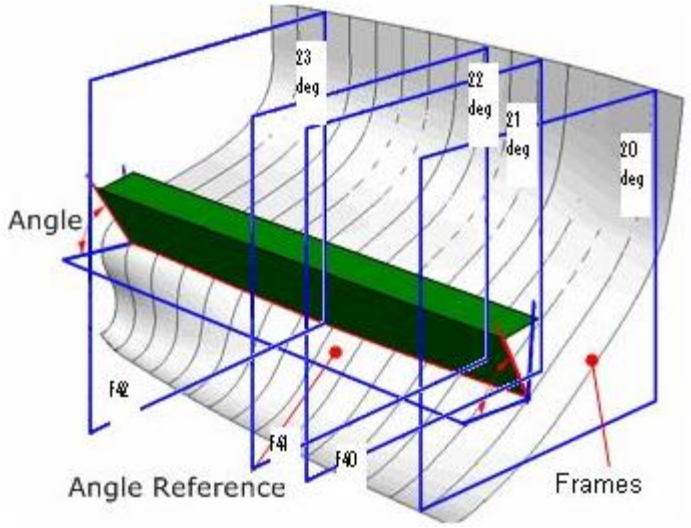
Twist is distributed using a cubic equation along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate at the ends is zero. The twist rate reaches its maximum at the midpoint.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.

- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Twist Linear Ortho

Twist is constant at every frame along the orthogonal projection length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The software uses the Main_CS frame system if it exists. Otherwise, the software uses the CS_0 frame system.



- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Profile System Table

Opens a read-only table containing the coordinates, orientation angles and twist rates of the current profile.

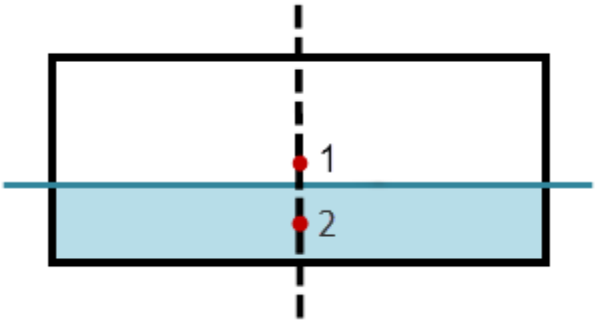
See Also

Molded Forms Workflow (on page 28)

Center of Buoyancy

The center of buoyancy of an object is located at the center of the volume of the liquid that the object displaces. It is the point through which the upward buoyant force acts.

The following diagram shows a rectangular form floating in water. The center of gravity (point 1) is at the geometric center of the shape. The center of buoyancy (point 2) is at the center of the volume of the displaced water.



1 - Center of Gravity

2 - Center of Buoyancy

For a ship, there is no single center of buoyancy value. Rather, there is a center of buoyancy for every floating position of the ship. This includes both non-heeled and heeled conditions up to and including the design draft (the maximum submerged floating position). Normally, calculations are done for a range of drafts at the 0° heeled state. These are reported in a tabular form or depicted graphically on a drawing along with other computed values. The collection of these values is commonly referred to as Hydrostatics.

Smart 3D does not directly calculate the center of buoyancy. The data used to compute the center of buoyancy of a ship is provided by the hull plate system. Because a ship is designed to meet conditions of performance (speed, carrying capacity, and so on) the hull form shape is usually developed before the three-dimensional model is started. Usually, the three-dimensional model is built based on preliminary hull designs.

Smart 3D allows you to use the third-party hull design tool that best meets your needs to create this preliminary design. You can then import that hull form as either an Initial Graphics Exchange Standard (IGES) file or a Standard ACIS Text (SAT) file. For more information, see *Import ACIS (File Menu)* (on page 1216) and *Import IGES (File Menu)* (on page 1216).

For floating offshore structures, there may be multiple hull plate systems, such as the pontoons and columns on semi-submersibles.

The tubular legs and braces of offshore jacket structures can be treated as hulls for floating calculations after the jacket is launched and positioned and before it is flooded. In these cases, you must know which structural objects are designed to be watertight and which are not. Smart 3D allows you to set properties on plate systems to indicate their tightness.

Smart 3D allows you to query structural system objects in the model database for the type (such as hull, deck, transverse bulkhead, longitudinal bulkhead, tube, and so on) as well as for their tightness (such as watertight, non-tight, weather tight, airtight, and so on). Such a query helps to identify which objects contribute to determining a center of buoyancy. These objects must be defined by the designer creating the model.

You must know what objects contribute to the floating position, the density of the fluid (salt or fresh water), the weight and center of gravity, and which objects are going to produce buoyant forces before you can calculate the center of buoyancy.

Calculations of this nature, and use of the computed data, are very specialized. They are done by professionals and experts in the field of naval architecture. Thicknesses of the structural members, such as pipes and tubes, have one volume when they are not flooded (volume of outer diameter) and another when they are flooded (volume of inner diameter). Submerged objects must take into account the volume of the structure itself because the net volume is what determines the actual buoyancy. Intergraph assumes no liability for use of data from Smart 3D for this purpose.

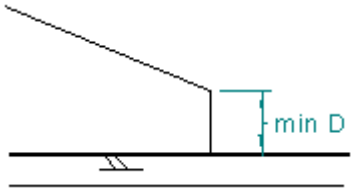
The three-dimensional model is built from early designs and it is rare to take a 3D model and try to compute these values. Intergraph recommends that you contact the group or company that was responsible for the design to get this type of data.

Minimum Distance

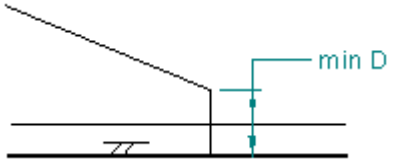
Minimum height or distances are maintained for plate systems with sketched boundaries and mother curves, profiles by projection, seams by projections, sketched openings, and stand-alone plate parts with sketched boundaries.

Minimum distances for sketched openings on plate systems are not supported.

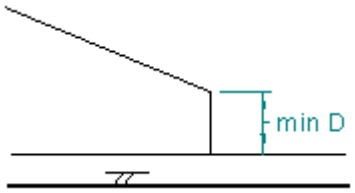
The software maintains the minimum height when the plate part is detailed in the Structural Detailing task. For example, if the sketched boundary is on the molded side of the bounding plate, then the minimum distance is maintained.



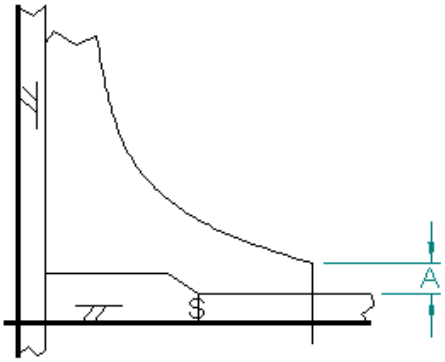
If the sketched boundary is on the thickness side of the bounding plate, then the sketched boundary is adjusted to compensate for thickness. For example, if the sketch is constrained to the molded side, the results are as follows:



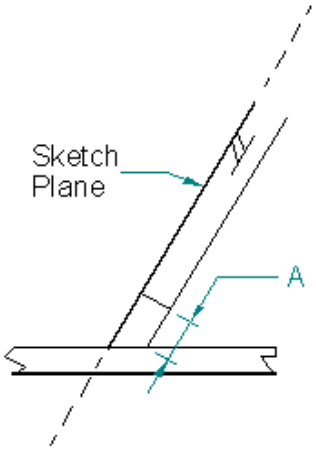
If the sketch is adjusted to the near side, the results would be as follows:



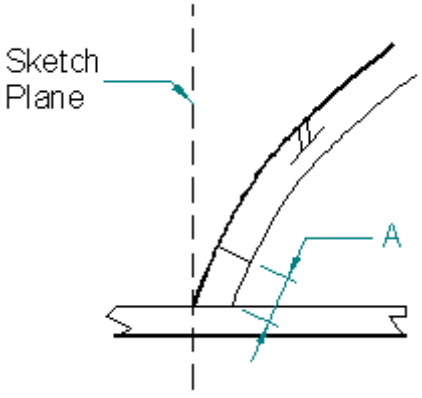
If the plate thickness is different along the length of the bounded plate, the minimum distance is applied using the plate thickness where the sketched boundary crosses the plate system. The results are as follows:



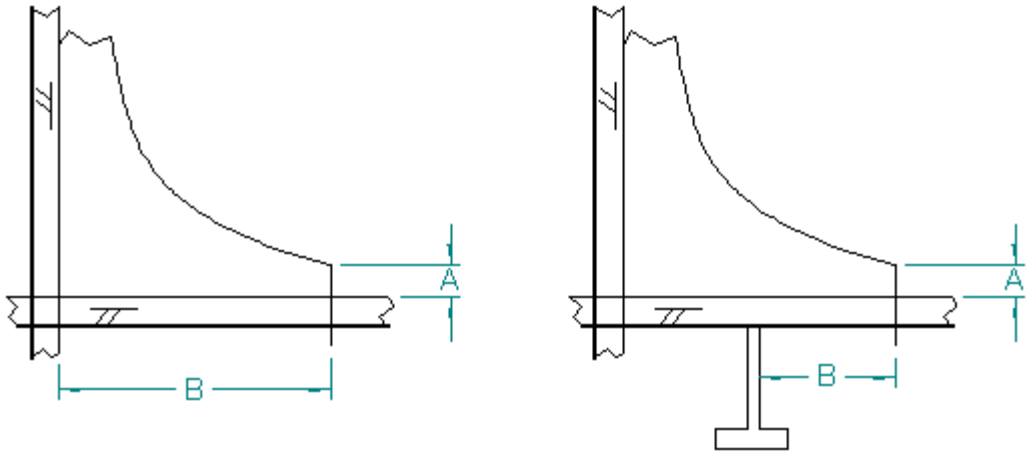
If the plate is defined with a sketched boundary that is not normal to the bounding plate, then the software addresses any long point - short point issues. The minimum distance is maintained on the shortest side, either the molded or anti- molded side.



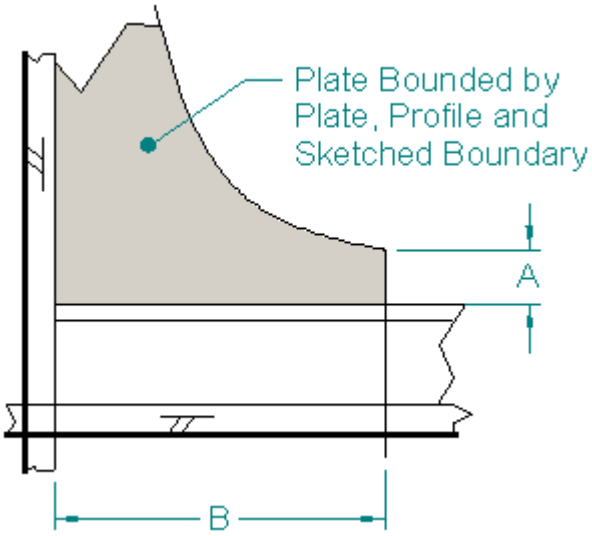
In non-planar cases, the minimum distance is maintained measured along the plate surface instead of in the sketch plane.



In cases where the sketched boundaries are constrained with several dimensions, the results are as follows:

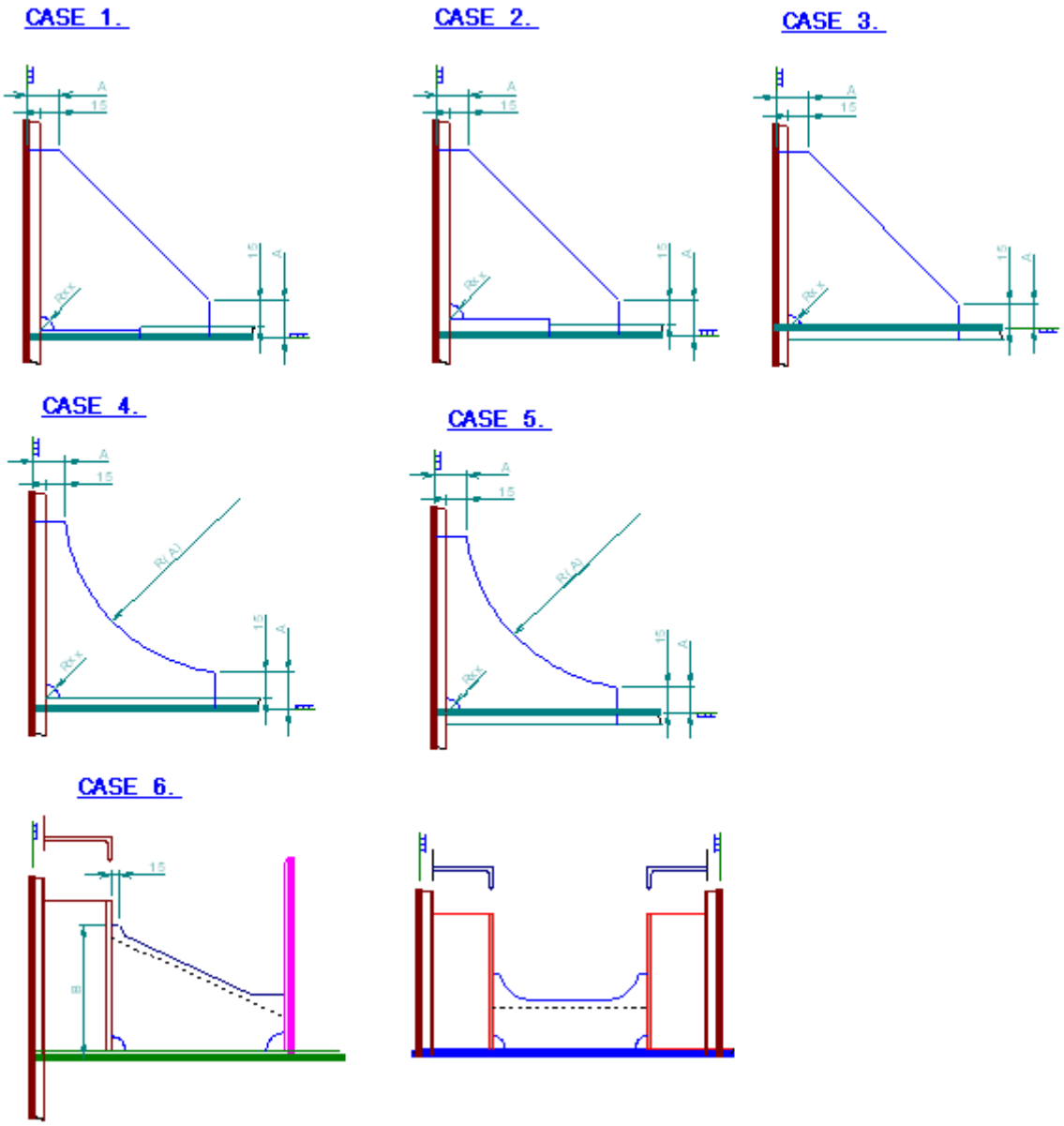


In cases where there is a sketched plate boundary plate bounded by a profile, the software resolves the long point - short point issues.



If there is a profile section size difference, the software maintains minimum dimensions where the sketched boundary crosses the profile system.

The following illustrations depict typical examples where minimum heights are maintained for sketched or symbolic tripping brackets. In these examples, **A** equals the base object thickness.



NOTE Minimum distance is controlled by a combination of Sketch2D and the symbol definition. You can identify dimensions that reflect minimum distances by adding graphic inputs in the symbol file. When the software finds these inputs in the symbol, then the software makes the necessary adjustments to ensure that the dimensions are maintained regardless of the supports thickness direction. For more information, see *2D Symbols User's Guide*.

SECTION 3

Placing Plate Systems

The Molded Forms task provides different groups of commands to create and represent single or multiple surfaces for plate systems. Also, you can represent and store imported surfaces. These commands start when you click the appropriate command icon on the vertical toolbar on the left side of the screen.



When you create any plate system, the first step is to use the **Properties** option to confirm the default attributes or define new values. Examples of attributes for a plate system include its name, category, molded conventions, its parent plate system, material properties, material thickness, tightness, and continuity.

The next step is defining the surface geometry of the plate system. You then define topological boundaries of the plate system by selecting objects in the three-dimensional workspace, selecting objects from the **Workspace Explorer**, or selecting objects from a list of potential bounding objects. For more information, see *Plate System Boundaries* (on page 98).

NOTES

- When a plate system is created, a plate part is also created. Plate parts are detailed in the Structural Detailing task and then used to create manufacturing parts in the Structural Manufacturing task. Properties of the part can be modified. For more information, see *Plate Part Properties Dialog Box* (on page 125).
- Minimum height or distances are maintained by the software for objects that use Sketch 2D, such as plate systems with sketched boundaries and mother curves, and standalone plate parts with sketched boundaries. For more information, see *Minimum Distance* (on page 92).
- When you create or edit a plate system or a stand-alone plate part, the dialog box title displays the plate system name using the format **[Plate System:** *<name of the plate system>***]**, as shown below.



Copying Plate and Profile Systems

When you copy, move, or mirror systems (including through **Model Data Reuse** and **Model Data Transform** operations in the Project Management task) without including a boundary in the copy (by selecting **Edit** > **Paste** > **Delete Optional**), the software replaces the boundary with a standalone boundary. This prevents the plate system from going to the **To Do List**. After the copy, you can replace the standalone boundary with a new boundary by using **Tools** > **Automatic Reconnect**.

If the original boundary was detailed, **Delete Optional** also deletes the physical connection between the plate system and the boundary. For the copied plate system, if you replace the

standalone boundary with a new detailed boundary, then the software creates a new physical connection.

For more information, see:

- *Automatic Reconnect (Tools Menu)* (on page 1388)
- *Move*, *Mirror Copy*, and *Paste Dialog Box* in the *Common User's Guide*
- *Model Data Reuse (MDR)* in the *Project Management User's Guide*
- *Model Data Transform (MDT)* in the *Project Management User's Guide*



Select - Used to select objects in the model. For more information, see *Selecting Objects* (on page 40).



Planar Plate System - Places a planar plate system. For more information, see *Planar Plate System* (on page 99).



Imported Plate System - Imports an ACIS or IGES file into the model. The file is used to create a plate system, typically a complex, non-planar system such as a ship hull. For more information, see *Imported Plate System* (on page 130).



Linear Extruded Plate System - Extrudes a plate system from a 2D curve normal to the sketching plane. For more information, see *Linear Extruded Plate System* (on page 156).



Nonlinear Extruded Plate System - Extrudes a plate system from a 2D curve along a linear or nonlinear curve that you specify. For more information, see *Nonlinear Extruded Plate System* (on page 180).



Revolved Plate System - Places a plate system by rotating a curve around an axis. For more information, see *Revolved Plate System* (on page 205).



Bracket Plate System - Creates a bracket plate system oriented by a plane or by its supports. For more information, see *Bracket Plate System* (on page 227).



Advanced Plate System - Creates specialized plate systems such as side plates, ring plates, transition plates, continuity plates, and buckling plates. For more information, see *Advanced Plate System* (on page 275).



Child Plate System - Creates a plate system that is a child to and in the same plane as another plate system. For more information, see *Child Plate System* (on page 426).



Ruled Plate System - Creates a plate system that is interpolated between two non-planar curves. For more information, see *Ruled Plate System* (on page 449).

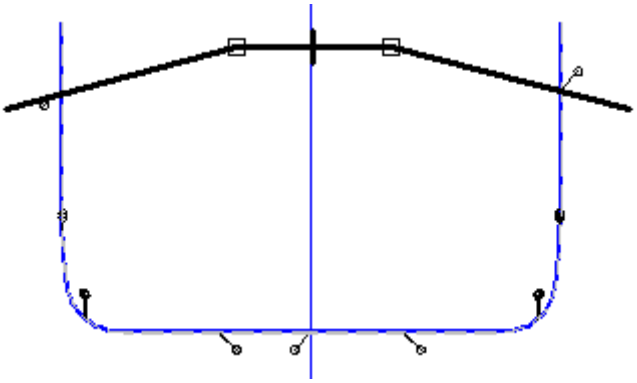
Plate System Boundaries

You can select any of the following as boundaries for plate systems:

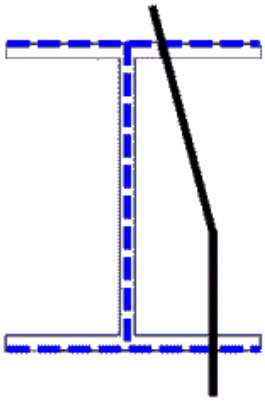
- Plate systems.
- Profile systems such as stiffeners, beams, or edge reinforcements. A plate can be bound to the webs and flanges of profile systems.
- Standard members created in the Structure task. A plate can be bound to the webs and flanges of members.
- Built-up members, but only when the web or flange plate systems of the built-up are selected.
- Sketched boundary curves.

Sketching Limitations

Sketched curves must extend beyond other bounding objects. In this example, the curve for a linear extruded deck extends beyond the hull boundary:



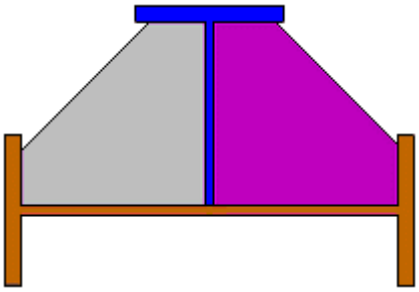
In this example, the bracket boundary curve extends beyond the top and bottom flanges:



Bounding and Splitting

A profile or member cannot be used to both bound and split a plate at the same time. In the following example, the two plates can be created in two different ways:

1. The plates are created separately, each using both members and a sketched line as boundaries.
2. A single plate is created using the horizontal member and two sketched lines as boundaries. After creation, the plate can then be split by the vertical member.



Planar Plate System

The **Planar Plate System** command creates planar plate systems. These plate systems are generally used to represent decks, sides of the deck house, transverse bulkheads, and longitudinal bulkheads, but are not limited to just those uses.



Creating a plate system involves three steps. First, define the plane of the plate system. Finally, define topological boundaries of the plate system. Then, use the **Properties** command to define attributes and defaults for the plate system. You can define the topological boundaries by selecting objects in a view, selecting objects from the **Workspace Explorer**, or selecting objects from a list of potential bounding objects.

Using Planar Plate Systems with Members

If a planar plate system is inserted along the length of a standard or built-up member, such as on the flange, the needed penetration logical connections, splits, and slot on the flange are not created automatically. You must then use **Manual Split** to create connections and splits and **Execute Detailing** to detail the plate and create the flange slot. The preferred method is to use **Advanced Plate System**, which automatically creates all needed connections, splits, and the flange slot. For more information, see *Advanced Plate System* (on page 275).

Planar Plate System Ribbon

Displays the options that you use to place the planar plate system in the model.

Plate System Properties

Opens the **Plate System Properties** dialog box, which you use to view and modify the properties of the plate system that you are about to place in the model. The initial plate system properties default from the selected structural specification. For more information, see *Planar Plate System Properties Dialog Box* (on page 106).

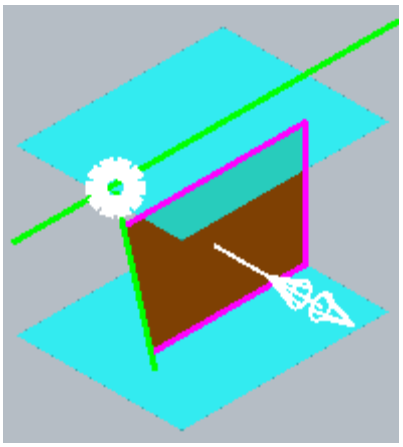
Define Plane

Specifies the reference plane of the plate system. You can select multiple planes to place multiple plate systems at the same time. The software uses the same boundaries for each plane and places the plate systems. After placement, the plate systems are unrelated to each other and can be edited as individual entities unless boundaries are defined as boundary curves. If so, then a single curve is related to all of the plate systems. See the *Define Plane Controls* section below.

Define Boundaries

Defines the outside boundaries of the plate system. You can define the boundaries by selecting objects, including structural members, in the model or by sketching boundary curves. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See the *Define Boundary Controls* and *Define Plane Controls* sections below.

NOTE When selecting an existing plate system, if displays for **Define Boundaries**, then the boundaries do not form a closed shape because a boundary has been modified. This condition most commonly occurs when the plate system has a sketched boundary that no longer intersects a plate system or reference plane boundary. Click **Define Boundaries** to display the location of the gap in the boundary.



Finish

Places the system using the defined parameters.

Define Boundary Controls

Available when you use the **Define Boundaries** option.

Pick Boundaries

Selects objects in the model or in the Workspace Explorer to define the boundaries. This option is only available when you are defining the boundaries.

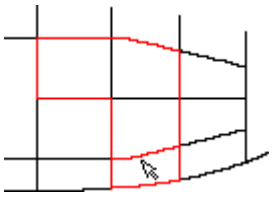
TIP After you select a boundary plate, press SHIFT to select the **Pick Edges** option. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Selects free edges on plates to define the boundary. This option is available when you select a plate in the **Pick Boundaries** step.

Solve Ambiguity

Clarifies the required boundary. If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to specify the boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Draws the boundaries. This option is only available when you are defining the boundaries. See *Define Plane Controls* below.

Sketching Plane

Selects the sketching plane for the boundary.

Add Intersecting Item

Allows you to select objects in the model that intersect the sketching plane to use as reference geometry. You see the selected objects in the Sketch 2D environment.

Add Projection Item

Allows you to select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the boundary.

Auto

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. The Workspace Explorer and the graphics view highlight items in the select set. If the software adds no objects to the select set, a message displays in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

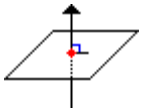
Define Plane Controls

Available when you use the **Define Plane** or **Boundary Curve** options.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only

available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

What do you want to do?

- *Place a planar plate system* (on page 104)
- *Modify a plate system* (on page 104)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a planar plate system

1. Click **Planar Plate System** on the vertical toolbar.
2. Click **Define Plane**.
3. Specify the plane for the plate system using one of the following methods:

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)
4. Define the plate system boundaries using one or more of the following methods:

   Pick boundaries (on page 81)
   Define a boundary curve (on page 82)
5. Click **Define Boundaries**.
6. If the boundary definition results in more than one solution, you must select bounded areas that together compose the total bounded area. Pick the needed solutions using the following procedure:

   Solve ambiguous solution created by selected boundaries (on page 82)
7. Set the remaining plate system properties.
8. Click **Finish**.

Modify a plate system

1. Click **Select** on the vertical toolbar.
2. Select **Molded Forms (Root Level)** in the **Locate Filter**.
3. Select the plate system to modify.

   The plate system ribbon appears.
4. Using the ribbon controls, edit the plate system as needed.

NOTES

- You cannot change the defining surface definition for plate systems that were created for a built-up member.
- You cannot delete individual plates from a built-up member.
- If you select a plate that is part of a built-up member, some of the boxes on the **Properties** dialog box are read-only.
- If the plate system is constrained to geometric construction points, you can modify the points. A separate ribbon displays for the points. For more information, see *Points (Geometric Construction Palette Dialog Box)* (on page 1028).

Move a plate system to a different system in the model hierarchy

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter**, select **Molded Forms (Root Level)**.
3. Select one or more root plate systems to move.

   The plate system ribbon appears.

   IMPORTANT
   - Select only the root plate system to move. The root plate system is not moved if any of its children in the hierarchy are also selected.
   - If you are selecting more than one root plate system, use CTRL+click to select only the root systems.
4. In the **Select Parent** box, select a new system. If the needed system is not available, click **More** to open the **Select System** dialog box.

   IMPORTANT If an invalid system, such as another root plate system, is selected, an error message displays and the plate system is not moved.
5. Click **Finish**.

   *The root plate system and its children move to the new system in the **Workspace Explorer**.*

NOTES

- Children of a root plate system (such as leaf plate systems and parts, profile systems and parts, openings, seams, and connections) always move with the root plate system.
- Children cannot be selected and moved out of the root plate system hierarchy.
- Valid systems are created in the Systems and Specifications task, or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.
- Root plate systems can also be moved in the Common task. The workflow and restrictions are similar, except that the system moves after making a selection in the **Select Parent** box because **Finish** is not available in the Common task.

Delete a plate system

1. Click **Select** on the vertical toolbar.
2. Select **Molded Forms (Root Level)** in the **Locate Filter**.
3. Select the plate system to delete.
4. Click **Delete**.

NOTE You cannot delete individual plates from a built-up member.

Planar Plate System Properties Dialog Box

Specifies the properties for the root or leaf plate system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Planar Plate System Properties Dialog Box) (on page 107)

Material Tab (Planar Plate System Properties Dialog Box) (on page 109)

Molded Conventions Tab (Planar Plate System Properties Dialog Box) (on page 110)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Main Tab (Planar Plate System Properties Dialog Box)

Specifies the general properties of the planar plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to

Deck. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

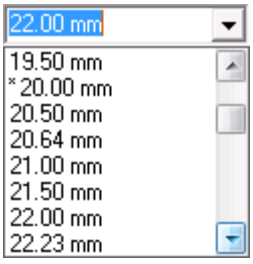
Specifies the level of tightness as it applies to the entire plate system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Planar Plate System Properties Dialog Box)

Specifies the material properties for the plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

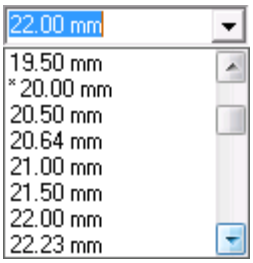
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Planar Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the plate system.

NOTE All profile orientation properties can only be modified at the root system.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



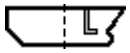
Starboard - Orients profile flanges to the starboard side.



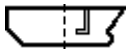
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



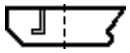
Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



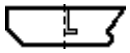
Aft - Orients profile flanges aft.



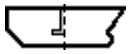
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



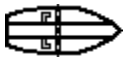
Aft - Orients profile flanges aft. This is the default value.



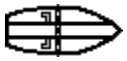
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

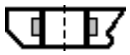
Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



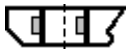
Outboard - Positions thickness away from the forward/aft reference.



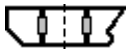
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

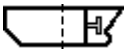
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



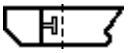
Aft - Orients profiles aft.



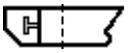
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



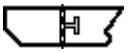
Aft - Orients profiles aft.



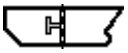
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



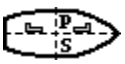
Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

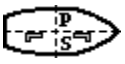
Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



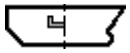
Starboard - Orients profile flanges to the starboard side.



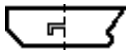
Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



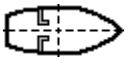
Aft - Orients profile flanges aft.



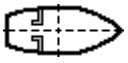
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



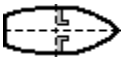
Aft - Orients profile flanges aft.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

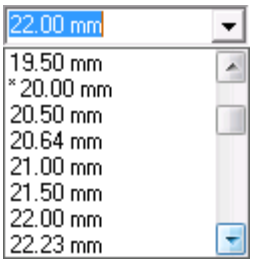


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.
- For plate and profile system part properties, parent systems are shown.
- For assembly connection properties, all connected objects are shown.
- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.
- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the

ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

General Tab

The **General** tab displays the properties that were selected by you or automatically determined by the software at creation. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

The properties displayed duplicate those available on other tabs, such as the **Main** tab. You can edit the properties on these tabs.

Plate Part Properties Dialog Box

Specifies the properties for the plate part that you are editing.

Main Tab (Plate Part Properties Dialog Box) (on page 125)
Material Tab (Plate Part Properties Dialog Box) (on page 127)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
General Tab (Plate Part Properties Dialog Box) (on page 127)
Weight & CG Tab (on page 129)
Extended User Attributes Tab (on page 129)
Routing Tab (on page 129)

Main Tab (Plate Part Properties Dialog Box)

Specifies the general properties of the plate part.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the object.

PlatePartRule - Creates a name based on the parent root plate system name, the location within the parent root plate system, and the parent assembly name.

User Defined - Name the plate part yourself by using the appropriate box.

Type

Displays the type of the plate part. It is inherited from the parent system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Displays the parent system for the object.

Surface Geometry Type

Displays the Molded Forms command used to create the root parent plate system. It is inherited from the parent system.

Specification

Specifies the specification for the object. This property is inherited from the parent system.

Description

Specifies a description for the object.

Tightness

Specifies the water tightness of the object. It is inherited from the parent system.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Material Tab (Plate Part Properties Dialog Box)

Displays the material properties for the plate part that you are editing. The material properties are inherited from the parent system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Actual Thickness

Displays the material thickness for the object.

General Tab (Plate Part Properties Dialog Box)

Specifies general properties of the plate part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Category: Standard

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Plate Part Properties Dialog Box)* (on page 125).

PlateTightness

Specifies the water tightness of the plate part. The list is defined by the StructPlateTightness codelist.

PlateType

Specifies the type of the plate part. The list is defined by the StructPlateType codelist.

Thickening Technique Requested

Specifies the technique requested to thicken the plate part.

This property affects only the behavior of plate parts derived from systems. The property defaults to **Procedural** at creation. Changing the value of this property can affect the geometry of the part as described below.

Standalone plate parts initially display nothing for this property. You can change the value, but those changes do not change the part or its geometry. Smart 3D thickens all standalone plate parts, including collars, using procedural thickening.

Undefined (blank) indicates that Smart 3D selects the appropriate technique. Smart 3D automatically changes this value to **Procedural**.

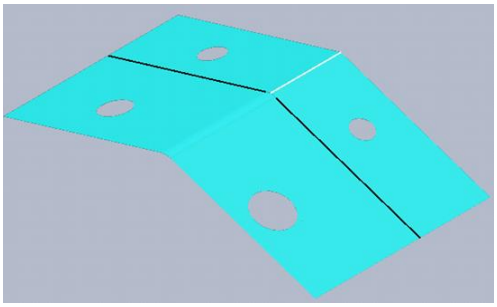
Procedural indicates the part is thickened using the standard thickening method provided by the ACIS modeler. This works for the vast majority of parts, and you should not change this value unless there is a specific reason to try a different technique.

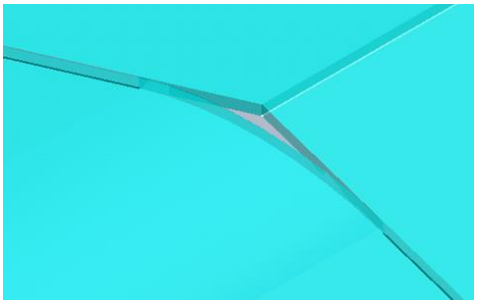
NOTE If the thickening fails using ACIS, and the part is a hull part, the software automatically tries the **NonProcedural** thickening technique. This option only controls which method the software tries first. You can determine which method Smart 3D actually used

from the **Thickening Technique Used** property.

Procedural with slit is used for parts that have internal slits or discontinuities in the surface of the system that form part of the boundary of parts on the system. The currently-supported example of this is a surface with a combination of split and bent knuckles joined end to end. Because Smart 3D does not support splitting knuckles and changing the properties of different segments, these surface must be constructed with the correct geometry to reflect the bent and split portions. Parts that are bounded by the slit, and potentially parts on the system that are within half a meter of the slit, must have this value set to assure a proper trim. Setting this attribute on parts that do not need it should not cause a failure but may cause the part to take more time to detail. The following pictures show an example of this type of surface, first the full surface and then a close-up in the region of the slit or discontinuity.

The plate system in these pictures is split into three parts by the knuckle (shown as white) and the two black seams. You must be sure that a single part does not touch both sides of the slit.





NonProcedural is an infrequently used option. Historically, a much higher percentage of parts failed to thicken with the ACIS modeler, and many designers used a specific hull form definition tool that produced a surface with very specific characteristics. **NonProcedural** thickening provided a tool that would thicken almost any part on this type of surface, even if the ACIS modeler failed. The ACIS modeler has improved greatly, and the **NonProcedural** thickening method has not been enhanced to handle the surfaces produced by many of the hull form tools currently in use. Setting this option may solve a failure-to-thicken problem on a hull part.

NOTE If **NonProcedural** thickening fails for a hull shell part, Smart 3D automatically tries to thicken the part using **Procedural** thickening. This option only controls which method the software tries first. You can determine which method Smart 3D actually used from the **Thickening Technique Used** property.

Thickening Technique Used

Displays the technique used to thicken the plate part. This is blank for a standalone plate part, although **Procedural** is always used for these parts. It displays one of the values defined for **Thickening Technique Requested** for a system-derived part. The meaning for these values is the same as defined for **Thickening Technique Requested**.

Category: Planning

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

Weight & CG Tab

Displays the center-of-gravity and the weight of the part.

Type

Displays the type of weight: **Dry Weight** or **Wet Weight**.

Weight

Displays the weight of the plate part pertaining to the weight type.

CoG X

Displays the position of center of gravity along the X-axis in the **CoG Coordinate System**.

CoG Y

Displays the position of center of gravity along the Y-axis in the **CoG Coordinate System**.

CoG Z

Displays the position of center of gravity along the Z-axis in the **CoG Coordinate System**.

CoG Coordinate System

Specifies the coordinate system to use to measure the center of gravity.

Extended User Attributes Tab

Displays the user attributes that have been bulkloaded on the report data object related to the part. Changes to these attributes do not cause the related manufacturing part to go out-of-date. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Routing Tab

Specifies routing properties for the part. By default, no routing properties are set.

Create

Sets routing properties for the part.

Delete

Deletes the routing properties for the part.

Rule

Specifies a predefined rule set or a user-defined rule that is applicable to the part.

Workcenter

Select an assembly workcenter where the part is produced from a facility-defined rule set. The list contains all available workcenters regardless of their level in the workcenter hierarchy. Examples of workcenters include docks, shops, bays, and panel lines. Workcenters may also be areas and zones where a block is assembled from multiple assemblies.

Stage Code

Specifies the workcenter code.

No. Of Actions

Specifies the number of actions required to produce the part.

Action

Specifies the name of the action.

Machine

Specifies the type of machine used to perform the required operation.

Code

Specifies the machine code.

Imported Plate System

 Imports an ACIS or IGES file into the model. The file is typically created using third-party applications such as Napa, Rhino, or TID. If the file represents a hull form, it can contain naval architecture reference curves.



Reference Curves

The software displays these curves as part of the plate system surface. The curves are also named, typed, and categorized, allowing you to query for a curve or select the curve and use it as a reference just like any other curve. You can modify the properties of reference curves, such as the name and type. You cannot modify the geometry of a naval architecture reference curve in the software. You must import a new file that contains the modified curve geometry.

The software supports the following ship curves that can be included as naval architecture references curves in the imported file:

- **Buttock_line** - A curve lying on the molded surface of a hull that is the intersection of a longitudinal plane with a hull molded form.

- **Centerline** - A curve lying on the molded surface of a hull that is the intersection of the longitudinal center plane with the hull molded form.
- **Flat_of_Bottom** - The boundary curve of the bottom planar surface at the hull molded form base.
- **Flat_of_Side** - The boundary curve of the side planar surface at the outer-most port or starboard side of the hull molded form.
- **Intersection_Line** - A curve that is the intersection of two surfaces found on, or within, a molded form.
- **Station_Line** - A curve lying on the hull molded surface that is the intersection of a transverse plane with a hull molded form.
- **Waterline** - A curve lying on the hull molded surface that is the intersection of the water plane with the hull molded form.
- **Bounding_Line** - Any bounding curve of a ship surface.
- **Unspecified** - A curve whose relation to naval architecture is not known or not recorded.

The software recognizes the following types of naval architecture reference curves:

- **Knuckle Curves (KNU)** - A 3D curve on the hull surface that exhibits tangent discontinuity (knuckle constraint) of more than one degree anywhere along its curve geometry. Multiple knuckle curves can occur and are given unique names. Knuckle reference curves support rules that determine the production method (bend, split, or ignore) governing their treatment and processing in the model.
- **Tangent Curves (TAN)** - A 3D curve on the hull surface that exhibits tangency constraint when intersected by orthogonal curves (section, waterlines, and buttocks). Multiple tangent curves can occur and are given unique names.
- **Stem and Stern Profiles Curves (STM PROF and STR PROF)** - The profile curves that define the outline of the hull at the Y=0 position. It is usually split into two individual curves, the Stem and Stern Profile. These two curves meet at amidships at Z=0. These curves are unique to single hulls.
- **Stem and Stern Half Siding Curves (HS STM and HS STR)** - The stem and stern half-siding curve is a 3D curve that defines the points at which waterline endings knuckle in to the centerline of the hull (stem and stern profile). The resulting surface bounded by the half-siding curve is a flat surface. These curves are unique to single hulls.
- **Stem and Stern Tangent Curves (STM TAN, STR TAN)** - The stem and stern tangent curve is a 3D curve that defines the points at which waterline endings are tangential into a circular or elliptical waterline ending. The resulting surface bounded by the tangent curve is a circular or elliptical part cone. These curves are unique in single hulls.

Imported Plate System Ribbon

Displays the options that you use to import a .SAT or .IGS file into the model.



Imported Plate System Properties

Opens the **Imported Plate System Properties** dialog box, which you use to view and modify the properties of the plate system that you are about to place in the model. The initial plate system properties default from the parent plate system, if there is one, or from the selected structural specification. For more information, see *Imported Plate System Properties Dialog Box* (on page 137).

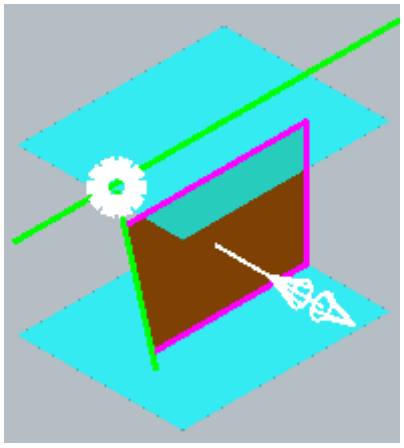
File Definition

Select the file to import. This option opens the **Open Hull File** dialog box.

Define Boundaries

Define the outside boundaries of the plate system. You can define the boundaries by selecting objects in the model or by drawing the boundaries. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See the *Define Boundary Controls* and *Define Plane Controls* sections below.

NOTE When selecting an existing plate system, if displays for **Define Boundaries**, then the boundaries do not form a closed shape because a boundary has been modified. This condition most commonly occurs when the plate system has a sketched boundary that no longer intersects a plate system or reference plane boundary. Click **Define Boundaries** to display the location of the gap in the boundary.



Finish

Places the system using the defined parameters.

Define Boundary Controls

These options are only available when you use the **Define Boundaries** option.

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

TIP Pressing the SHIFT key selects the **Pick Edges** option after you have selected a boundary plate. Press SHIFT again to return to the **Pick Boundaries** option.

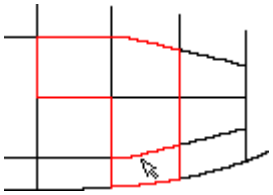
Pick Edges

Select this option to pick free edges on plates to define the boundary. This option is available if you selected a plate in the **Pick Boundaries** step.

Solve Ambiguity

If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple

bounded areas to clarify the needed boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Select this option to draw the boundaries. This option is only available when you are defining the boundaries.

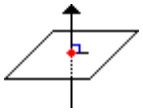
Define Plane Controls

Only available when you use the **Boundary Curve** option.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see

Select System Dialog Box in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.



Mirror Surface

Mirrors the surface about the longitudinal centerline. This option is only available when you are importing a plate from an .IGS file. You can use this option for a complete hull surface when the IGES file defines only half of the hull. A preview for the hull is updated automatically when you select or clear the option. The default value is not to mirror and is reset to this state after you click **Finish**.



Mirror Reference Curves

Mirrors the reference curves about the longitudinal centerline. This option is only available when you are importing a plate from an .IGS file. You can use this option for complete reference curves when the IGES file defines only half of the hull. The default value is not to

mirror and is reset to this state after you click **Finish**.

What do you want to do?

- *Import a plate system* (on page 136)
- *Swap hulls* (on page 137)
- *Modify a plate system* (on page 104)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Import a plate system

This workflow applies to all imported plates systems, including hulls.

1. On the **Molded Forms** vertical toolbar, click **Imported Plate System** .
2. Select the .SAT or .IGS file to import.

   NOTE To show .IGS files in the file browser, you must select **IGES files (*.igs)** as the file type.
3. Click **Open**.
4. If you are importing a hull from an .IGS file and the file defines only half of the hull, click **Mirror surface** , **Mirror reference curves** , or both to import a complete hull.

   NOTE By default, the two mirror options are disabled. A preview for the hull is updated automatically when you click **Mirror surface** .
5. On the ribbon bar, select an appropriate plate type. For example, select **Hull** for a hull form.
6. Specify the properties for the imported plate system.
7. Define the boundaries for the imported plate system.
8. Click **Finish**.

NOTES

- A complete hull form may not have boundaries.
- If you define an ambiguous boundary, you must provide more information by selecting bounded areas that together compose the total bounded area.
- For a hull imported from IGES, an import log file and a .SAT file are automatically created in the same folder as the source .IGS file. The surfaces and curves are automatically named based on the data in the .IGS file directory, and all imported curves are set to **Reference** type. You can modify the curve type of multiple curves by selecting the reference curves in the **Workspace Explorer** and then clicking **Reference Curve Properties** .
- A minimum set of checks are performed on the data to ensure that it meets the criteria necessary to successfully bound, trim, detail, and manufacture in the software. These checks are the same as those available in the Geometry Analysis and Repair (GAR) task. If any of the checks are not met, then the plate system is created, but it is added to the **To Do List** with an appropriate error message. You must review the data in GAR, determine the cause of

the problem, and make needed corrections. The third-party tool used to create the geometry might be needed to make the corrections.

Swap hulls

Follow this procedure only if you are very early in the design cycle with a model having a minimal number of objects detailed in the Structural Detailing task. Otherwise, follow the procedure described in *Swap Hulls on a Detailed Model Using Molded Forms Delay* (on page 1396).

1. Click **Select** on the vertical toolbar.
2. Select **Molded Forms (Root Level)** in the **Locate Filter**.
3. Select the existing hull in the model.
4. On the ribbon, select **File Definition**.
5. Select the .SAT or .IGS file that contains the new hull to import.

   NOTE To show .IGS files in the file browser, you must select **IGES files (*.igs)** as the file type.
6. Click **Open**.
7. If you are importing a hull from an .IGS file and the file defines only half of the hull, click **Mirror surface**, **Mirror reference curves**, or both to import a complete hull.

   NOTE By default, the two mirror options are disabled. A preview for the hull is updated automatically when you click **Mirror surface**
8. Click **Finish**.

Imported Plate System Properties Dialog Box

Specifies the properties for the root or leaf imported plate system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.

- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Imported Plate System Properties Dialog Box) (on page 138)

Material Tab (Imported Plate System Properties Dialog Box) (on page 140)

Molded Conventions Tab (Imported Plate System Properties Dialog Box) (on page 141)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Delay Tab (Imported Plate System Properties Dialog Box) (on page 155)

Main Tab (Imported Plate System Properties Dialog Box)

Specifies the general properties of the imported plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: <*Reference plane name*>-<*Index number*><*Plate type*>-<*Workshare location ID*>. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): <*Global CS axis*><*Global CS position along the axis in mm*>-<*Index number*><*Plate type*>-<*Workshare location ID*>, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A*-<*Unique index number*><*Plate type*>-<*Workshare location ID*>, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

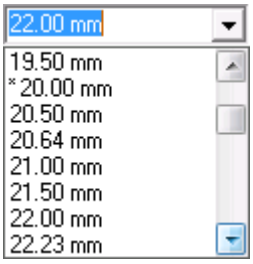
Specifies the level of tightness as it applies to the entire plate system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Imported Plate System Properties Dialog Box)

Specifies the material properties for the plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

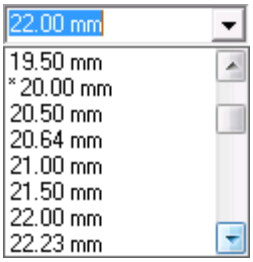
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Imported Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the plate system that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



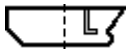
Starboard - Orients profile flanges to the starboard side.



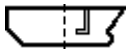
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



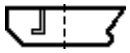
Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



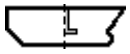
Aft - Orients profile flanges aft.



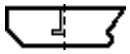
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



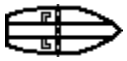
Aft - Orients profile flanges aft. This is the default value.



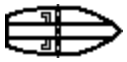
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.

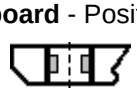


Transverse Bulkhead Molded Conventions

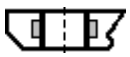
Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



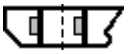
Outboard - Positions thickness away from the forward/aft reference.



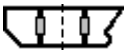
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

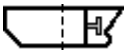
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



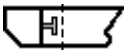
Aft - Orients profiles aft.



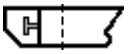
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



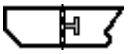
Aft - Orients profiles aft.



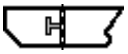
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



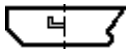
Starboard - Orients profile flanges to the starboard side.



Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



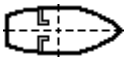
Aft - Orients profile flanges aft.



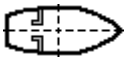
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



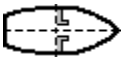
Aft - Orients profile flanges aft.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the

plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for

designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

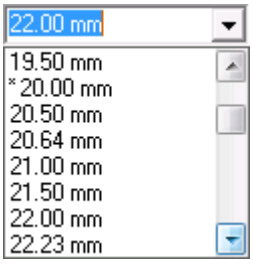


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Delay Tab (Imported Plate System Properties Dialog Box)

Specifies properties for delaying recalculation of the imported plate system that you are editing.

Delay Settings

Molded Forms Delay - Delays recalculation of Molded Forms objects. The objects are instead added to the To Do List.

Plate Split Delay - Delays splitting of the plate system by seams.

Range Settings

User range of change - Restricts recalculation to one or more range boxes that you specify. Type low and high values for X, Y, and Z in **User Range** to create a box.

System range of change - Restricts recalculation to a range box created by the software and based on the X, Y, and Z differences between the old and new imported plate systems.

User Range

Low - Specifies the lowest value along a coordinate system axis. Type values for **X**, **Y**, and **Z**.

High - Specifies the highest value along a coordinate system axis. Type values for **X**, **Y**, and **Z**.

Modify - For a selected existing row in the **Display Range** grid, changes the **Low** and **High** values to the new values.

Add - Adds a new row of **Low** and **High** values to the **Display Range** grid.

Display Range - Displays range boxes where recalculation is delayed. Each row in the grid contains low and high coordinate values for a range box.

☐ - Selects a row in the **Display Range** grid. When a row is selected, the range box displays in a graphic view.

Clear All - Clears row selection in **Display Range**.

Select All - Selects all rows in **Display Range**.

Delete - Removes the selected row from **Display Range**.

Delete All - Removes all rows from **Display Range**.

OK - Accepts the values and closes the dialog box.

Cancel - Rejects the values and closes the dialog box.

Apply - Accepts the values but does not close the dialog box.

Linear Extruded Plate System

The **Linear Extruded Plate System** command extrudes a plate system from a 2D curve normal to the sketching plane. You define the 2D curve that the software uses to define the plate system surface geometry.



TIP Use the **Nonlinear Extruded Plate System** command if you need the 2D curve to follow a curved path instead of always being normal to the sketching plane. For more information, see *Nonlinear Extruded Plate System* (on page 180).

Plate Knuckles

If the plate system has an angle with no radius on the extrusion curve, the software creates a plate knuckle object. You can select this object graphically, or in the **Workspace Explorer**. For more information, see *Plate Knuckle Properties Dialog Box* (on page 931).

Linear Extruded Plate System Ribbon

Displays the options that you use to extrude a plate system along a linear extrusion curve in the model.

Plate System Properties

Opens the **Linear Extruded Plate System Properties** dialog box, which you use to view and modify the properties of the plate system. The initial plate system properties default from the parent plate system, if there is one, or from the selected structural specification. For more information, see *Linear Extruded Plate System Properties Dialog Box* (on page 162).

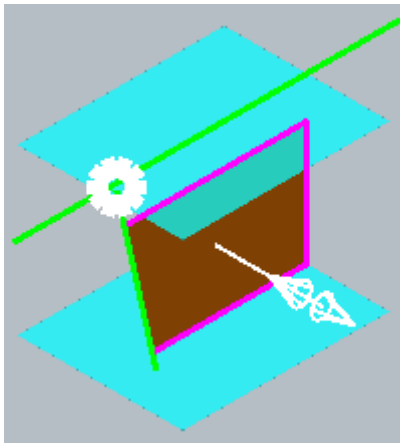
Mother Curve Definition

Defines the sketching plane and the shape of the mother curve. See the *Define Mother Curve Controls* and *Define Plane Controls* sections below.

Define Boundaries

Specifies the outside boundaries of the plate system. You can define the boundaries by selecting objects in the model or by drawing the boundaries. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See the *Define Boundary Controls* and *Define Plane Controls* sections below.

NOTE When selecting an existing plate system, if  displays for **Define Boundaries**, then the boundaries do not form a closed shape because a boundary has been modified. This condition most commonly occurs when the plate system has a sketched boundary that no longer intersects a plate system or reference plane boundary. Click **Define Boundaries**  to display the location of the gap in the boundary.



Finish

Places the system using the defined parameters.

Define Mother Curve Controls

Available when you use the **Mother Curve Definition** option.

Sketching Plane

Specify the sketching plane for the curve. This is the first step in defining the curve. See the *Define Plane Controls* section below.



Add Intersecting Item

Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.



Add Projection Item

Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.



Sketch 2D

Opens the Sketch 2D environment in which you can draw the curve.

NOTE To display a grid or reference plane ruler in the Sketch 2D environment, first define an active coordinate system using **Tools** > **PinPoint**. For more information, see PinPoint in the *Common User's Guide* and *2D Grids Ruler* in the *SmartSketch Drawing Editor Help*.

Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a

message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Project Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Define Boundary Controls

Available when you use the **Define Boundaries** option.

Pick Boundaries

Selects objects in the model or in the Workspace Explorer to define the boundaries. This option is only available when you are defining the boundaries.

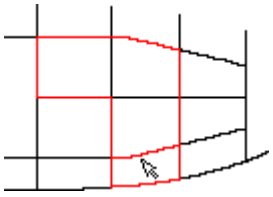
TIP After you select a boundary plate, press SHIFT to select the **Pick Edges** option. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Selects free edges on plates to define the boundary. This option is available when you select a plate in the **Pick Boundaries** step.

Solve Ambiguity

Clarifies the required boundary. If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to specify the boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Draws the boundaries. This option is only available when you are defining the boundaries. See *Define Plane Controls* below.

Sketching Plane

Selects the sketching plane for the boundary.

Add Intersecting Item

Allows you to select objects in the model that intersect the sketching plane to use as reference geometry. You see the selected objects in the Sketch 2D environment.

Add Projection Item

Allows you to select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the boundary.

Auto

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. The Workspace Explorer and the graphics view highlight items in the select set. If the software adds no objects to the select set, a message displays in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

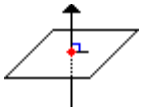
Define Plane Controls

Available when you use the **Mother Curve Definition** or **Boundary Curve** options.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

/ Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube**

Plate, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

What do you want to do?

- *Place a linear extruded plate system* (on page 161)
- *Modify a plate system* (on page 104)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a linear extruded plate system

1. Click **Linear Extruded Plate System** on the vertical toolbar.
2. Specify the sketching plane for the mother curve using one of the following methods:

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)

3. Click **Add Intersecting Item**  .
4. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.
5. Click **Add Projection Item**  .
6. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.
7. Click **Sketch 2D**  .
8. In the Sketch 2D environment, sketch the mother curve for the extruded plate system.
9. Click **Finish** in the Sketch 2D environment.
10. Click **Define Boundaries**  .
11. Define the plate system boundaries using one or more of the following methods:

    Pick boundaries (on page 81)

    Define a boundary curve (on page 82)
12. Click **Accept**  .
13. If the boundary definition results in more than one solution, you must select bounded areas that together compose the total bounded area. Pick the needed solutions using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
14. Set the remaining plate system properties.
15. Click **Finish**.

Linear Extruded Plate System Properties Dialog Box

Specifies the properties for the root or leaf plate system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.

- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Linear Extruded Plate System Properties Dialog Box) (on page 163)

Material Tab (Linear Extruded Plate System Properties Dialog Box) (on page 166)

Molded Conventions Tab (Linear Extruded Plate System Properties Dialog Box) (on page 166)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Main Tab (Linear Extruded Plate System Properties Dialog Box)

Specifies the general properties of the linear extruded plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as

Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

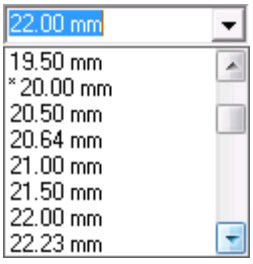
Specifies the level of tightness as it applies to the entire plate system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Linear Extruded Plate System Properties Dialog Box)

Specifies the material properties for the linear extruded plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

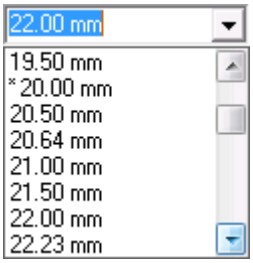
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Linear Extruded Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the linear extruded plate system that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the

longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



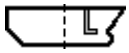
Starboard - Orients profile flanges to the starboard side.



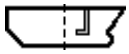
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



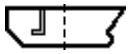
Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



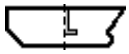
Aft - Orients profile flanges aft.



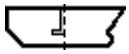
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



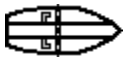
Aft - Orients profile flanges aft. This is the default value.



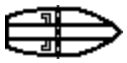
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



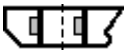
Outboard - Positions thickness away from the forward/aft reference.



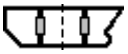
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

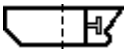
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



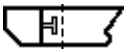
Aft - Orients profiles aft.



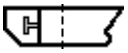
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



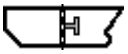
Aft - Orients profiles aft.



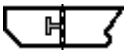
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



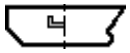
Starboard - Orients profile flanges to the starboard side.



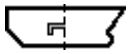
Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



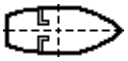
Aft - Orients profile flanges aft.



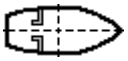
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



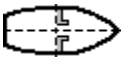
Aft - Orients profile flanges aft.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the

opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface

normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the

opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

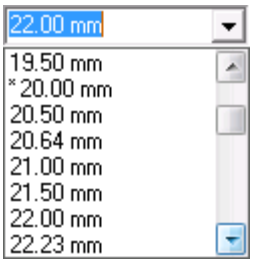


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Nonlinear Extruded Plate System

The **Nonlinear Extruded Plate System** command extrudes a plate system from a 2D curve along a linear or nonlinear curve that you specify. The software uses the 2D curves to define the plate system surface geometry.



TIP Use the **Linear Extruded Plate System** command instead of this command if the 2D curve remains normal to the sketching plane. For more information, see *Linear Extruded Plate System* (on page 156).

Plate Knuckles

If the plate system has an angle with no radius on the extrusion curve, the software creates a plate knuckle object. You can select this object graphically, or in the **Workspace Explorer**. For more information, see *Plate Knuckle Properties Dialog Box* (on page 931).

Nonlinear Extruded Plate System Ribbon

Displays the options that you use to extrude a plate system along a non- linear extrusion curve in the model.

Plate System Properties

Opens the **Nonlinear Plate System Properties** dialog box, which you use to view and modify the properties of the plate system that you are about to place in the model. The initial plate system properties default from the parent plate system, if there is one, or from the selected structural specification. For more information, see *Nonlinear Plate System Properties Dialog Box* (on page 187).

Extrusion Definition

Specify the path the mother curve is to follow. See the *Define Curve Controls* and *Define Plane Controls* sections below.

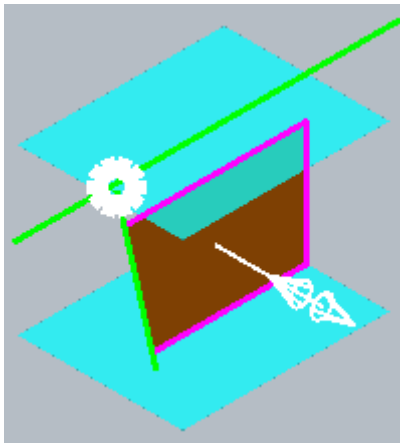
Mother Curve Definition

Defines the sketching plane and the shape of the mother curve. See the *Define Curve Controls* and *Define Plane Controls* sections below.

Define Boundaries

Defines the outside boundaries of the plate system. You can define the boundaries by selecting objects in the model or by drawing the boundaries. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See the *Define Boundary Controls* and *Define Plane Controls* sections below.

NOTE When selecting an existing plate system, if displays for **Define Boundaries**, then the boundaries do not form a closed shape because a boundary has been modified. This condition most commonly occurs when the plate system has a sketched boundary that no longer intersects a plate system or reference plane boundary. Click **Define Boundaries** to display the location of the gap in the boundary.



Finish

Places the system using the defined parameters.

Define Curve Controls

Available when you use the **Extrusion Definition** and **Mother Curve Definition** options.

Sketching Plane

Specify the sketching plane for the curve. This is the first step in defining the curve. See the *Define Plane Controls* section below.

Add Intersecting Item

Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.

Add Projection Item

Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the curve.

NOTE To display a grid or reference plane ruler in the Sketch 2D environment, first define an active coordinate system using **Tools** > **PinPoint**. For more information, see PinPoint in the *Common User's Guide* and *2D Grids Ruler* in the *SmartSketch Drawing Editor Help*.

Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Project Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Define Boundary Controls

Available when you use the **Define Boundaries** option.

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

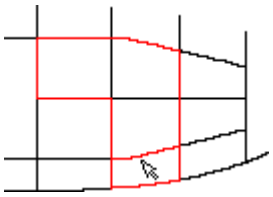
TIP Pressing the SHIFT key selects the **Pick Edges** option after you have selected a boundary plate. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Select this option to pick free edges on plates to define the boundary. This option is available if you selected a plate in the **Pick Boundaries** step.

Solve Ambiguity

If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to clarify the needed boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Select this option to draw the boundaries. This option is only available when you are defining the boundaries.

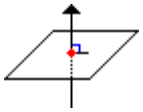
Define Plane Controls

Available when you use the **Extrusion Definition**, **Mother Curve Definition** or **Boundary Curve** options.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable

options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

What do you want to do?

- *Place a nonlinear extruded plate system* (on page 185)
- *Modify a plate system* (on page 104)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a nonlinear extruded plate system

1. Click **Nonlinear Extruded Plate System** on the vertical toolbar.
2. Specify the grid plane for the extruded curve using one of the following methods:

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)
3. Click **Add Intersecting Item** .
4. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.
5. Click **Add Projection Item** .

6. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.
7. Click **Sketch 2D** .
8. In the Sketch 2D environment, sketch the extruded curve.

   TIP Be sure to read *Sketch 2D Best Practices* (on page 65).
9. Click **Finish** in the Sketch 2D environment.
10. Click **Mother Curve Definition** .
11. Specify the sketching plane for the mother curve using one of the following methods:

    Define a coincident plane (on page 76)
    Define an offset plane (on page 77)
    Define a plane using a point and a normal vector (on page 78)
    Define a plane using three points (on page 79)
12. Click **Add Intersecting Item** .
13. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.
14. Click **Add Projection Item** .
15. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.
16. Click **Sketch 2D** .
17. In the Sketch 2D environment, sketch the mother curve for the extruded plate system.
18. Click **Finish** in the Sketch 2D environment.
19. Click **Define Boundaries** .
20. Define the plate system boundaries using one or more of the following methods:

    Pick boundaries (on page 81)

    Define a boundary curve (on page 82)
21. Click **Accept** .
22. If the boundary definition results in more than one solution, you must select bounded areas that together compose the total bounded area. Pick the needed solutions using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
23. Set the remaining plate system properties.
24. Click **Finish**.

Nonlinear Plate System Properties Dialog Box

Specifies the properties for the root or leaf nonlinear extruded plate system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Nonlinear Plate System Properties Dialog Box) (on page 187)

Material Tab (Nonlinear Plate System Properties Dialog Box) (on page 190)

Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box) (on page 191)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Main Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the general properties of the nonlinear extruded plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type

(Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.

- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the

property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

Specifies the level of tightness as it applies to the entire plate system.

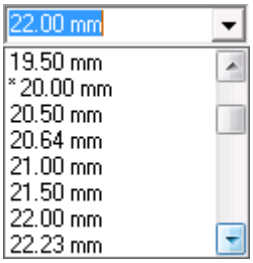
Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on

page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the material properties for the nonlinear extruded plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

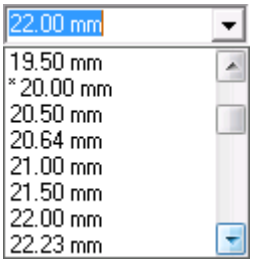
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the nonlinear extruded plate system that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



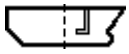
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



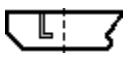
Aft - Orients profile flanges aft.



Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



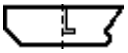
Aft - Orients profile flanges aft.



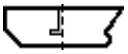
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



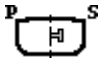
Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



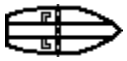
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



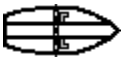
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



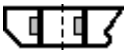
Outboard - Positions thickness away from the forward/aft reference.



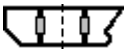
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

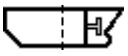
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



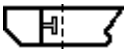
Aft - Orients profiles aft.



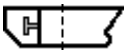
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



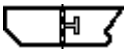
Aft - Orients profiles aft.



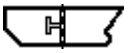
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



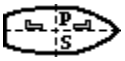
Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

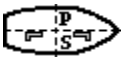
Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



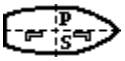
Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



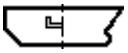
Starboard - Orients profile flanges to the starboard side.



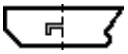
Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



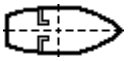
Aft - Orients profile flanges aft.



Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



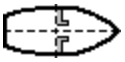
Aft - Orients profile flanges aft.



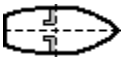
Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube

member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

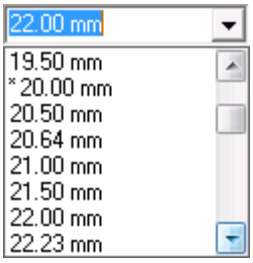


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Revolved Plate System

The **Revolved Plate System** command creates a plate system by rotating a curve around an axis. You define the 2D curve that the software uses to define the plate system surface geometry and the axis of revolution.



Revolved Plate System Ribbon

Displays the options that you use to place a rotated plate system in the model.

Plate System Properties

Opens the **Revolved Plate System Properties** dialog box, which you use to view and modify the properties of the plate system that you are about to place in the model. The initial plate system properties default from the parent plate system, if there is one, or from the selected structural specification. For more information, see *Revolved Plate System Properties Dialog Box* (on page 210).

First Curve Definition

Defines the shape of the mother curve and the axis of rotation. See the *Define Curve Controls*, *Sketch 2D Command*, and *Define Plane Controls* sections below.

Second Curve Definition

Defines the extent of the revolution for the mother curve about the axis of rotation. See the *Define Revolution Controls* section below.

Define Boundaries

Defines the outside boundaries of the plate system. You can define the boundaries by selecting objects in the model or by drawing the boundaries. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See the *Define Plane Controls* sections below.

Finish

Places the system using the defined parameters.

Define Curve Controls

Available when you use the **First Curve Definition** option.

Sketching Plane

Specify the sketching plane for the curve. This is the first step in defining the curve. See the *Define Plane Controls* section below.

Add Intersecting Item

Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.

Add Projection Item

Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the curve.

NOTE To display a grid or reference plane ruler in the Sketch 2D environment, first define an active coordinate system using **Tools** > **PinPoint**. For more information, see PinPoint in the *Common User's Guide* and *2D Grids Ruler* in the *SmartSketch Drawing Editor Help*.

Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Project Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Sketch 2D Command

Axis of Rotation

Select a line element to designate as the axis of rotation. The line is green when selected. This **Axis of Rotation** command is available only with the **First Curve Definition** option. It displays on the **Draw** toolbar of the **Sketch 2D** environment.

Define Revolution Controls

Available when you use the **Second Curve Definition** option.

360° Revolution

Select to have the mother curve rotate 360 degrees around the axis of rotation.

Revolution Angle

Select to limit the mother curve rotation about the axis of rotation.

Angle

Type the angle for the mother curve to rotate about the axis of rotation. You must define the direction by clicking the mouse button when the cursor is on the side in which to rotate. This option is only available when the **Revolution Angle** option is selected.

Both Directions

Select to have the mother curve rotate both directions about the axis of rotation from the sketching plane used to define the mother curve. The mother curve is rotated by the **Angle** value in each direction.

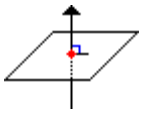
Define Plane Controls

Available when you use the **First Curve Definition** or **Define Boundaries** options.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available

when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

What do you want to do?

- *Place a revolved plate system* (on page 209)
- *Modify a plate system* (on page 104)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a revolved plate system

1. Click **Revolved Plate System** on the vertical toolbar.
2. Specify the sketching plane for the curve and rotation axis using one of the following methods:

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)
3. Click **Add Intersecting Item**.
4. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.
5. Click **Add Projection Item**.
6. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.
7. Click **Sketch 2D**.
8. In the Sketch 2D environment, sketch the curve for the revolved plate system. This is the curve that is to revolve around the axis.
9. Draw a straight line to represent the axis of rotation for the revolved plate system.
10. Select **Axis of Rotation** (located under the **Select** command in the Sketch 2D environment), and then select the straight line that you drew for the axis of rotation. The software changes the line color to green to indicate that it accepts the selected line as the axis of rotation.
11. Click **Finish** in the Sketch 2D environment.
12. Click **Accept**.
13. Specify the revolution angle.

14. Click **Accept**.
15. Define the plate system boundaries using one or more of the following methods:

    Pick boundaries (on page 81)
    Define a boundary curve (on page 82)
16. Click **Accept**.
17. If the boundary definition results in more than one solution, you must select bounded areas that together compose the total bounded area. Pick the needed solutions using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
18. Set the remaining plate system properties.
19. Click **Finish**.

Revolved Plate System Properties Dialog Box

Specifies the properties for the root or leaf plate system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Revolved Plate System Properties Dialog Box) (on page 211)

Material Tab (Revolved Plate System Properties Dialog Box) (on page 213)

Molded Conventions Tab (Revolved Plate System Properties Dialog Box) (on page 214)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Main Tab (Revolved Plate System Properties Dialog Box)

Specifies the general properties of the revolved plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to

Deck. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

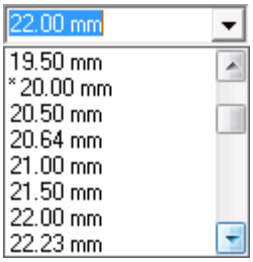
Specifies the level of tightness as it applies to the entire plate system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Revolved Plate System Properties Dialog Box)

Specifies the material properties for the revolved plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

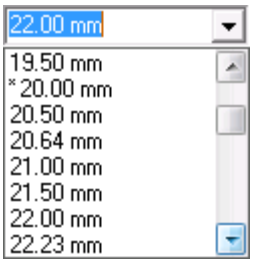
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Revolved Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the revolved plate system that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



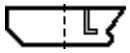
Starboard - Orients profile flanges to the starboard side.



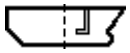
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



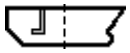
Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



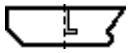
Aft - Orients profile flanges aft.



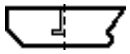
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



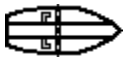
Aft - Orients profile flanges aft. This is the default value.



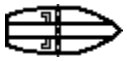
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

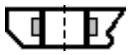
Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



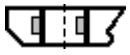
Outboard - Positions thickness away from the forward/aft reference.



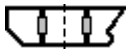
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

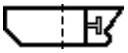
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



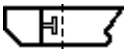
Aft - Orients profiles aft.



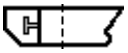
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



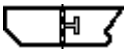
Aft - Orients profiles aft.



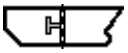
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



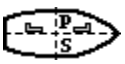
Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

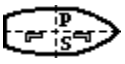
Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



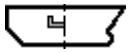
Starboard - Orients profile flanges to the starboard side.



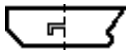
Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



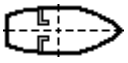
Aft - Orients profile flanges aft.



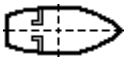
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



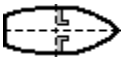
Aft - Orients profile flanges aft.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

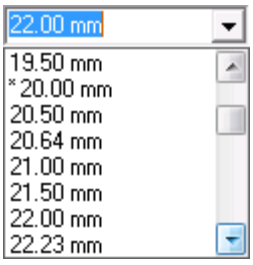


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Bracket Plate System

 Creates plate systems for primary structural brackets that can be stiffened, penetrated by stiffeners on the bounding structure, and split with design seams.

A bracket plate system is always bounded to the structure that it supports. You select a plane to locate and orient the bracket. The bracket has predefined shapes defined by symbols and rules.

A bracket is classified by its plane definition and its number of supports. Brackets are oriented by a selected plane or by the supports.

	Bracket by Plane	**Bracket by Supports**
Bracket plane orientation defined by:	Reference plane.	The plane of one of the supports (profile web or plate).
Types of supports	Plate system, profile system web, profile system flange, member system web, and member system flange.	Plate system and profile system.
Plane controls used	By Plane or Offset Plane Bracket Plane by Point and Vector Plane by Three Points	Plane by Selected Supports
Use the command when:	Supports cannot be used to orient the bracket. For more information, see *Bracket by Plane* (on page 246).	Supports can be used to orient the bracket. For more information, see *Bracket by Supports* (on page 247).

Bracket Plate System Ribbon

Displays the controls used to place bracket plate systems.

Properties

Opens the **Bracket System Properties** dialog box, which you use to view and modify the properties of the bracket system that you are about to place in the model. The initial bracket properties default from the selected structural specification and the bracket rules. For more information, see *Bracket System Properties Dialog Box* (on page 250).

Bracket Browser

Opens the **Define Bracket Plate System Properties** dialog box that specifies bracket symbols, parameters, and properties in an icon-based browser. You can select a symbol for the bracket prior to choosing the supports. For more information, see *Define Bracket Plate System Properties Dialog Box* (on page 272).

Bracket

Displays the bracket symbol. You can also manually select a bracket symbol in the catalog when **Select from the Catalog** is active.

Select from the Catalog

Allows the selection of any bracket symbol in the catalog in the **Bracket** box. This option is a toggle with **Create by Rule**.

Create by Rule

Limits the **Bracket** box to applicable rule-based symbols. This is the default option. This option is a toggle with **Select from the Catalog** .

Rule Based - When **Create by Rule** is active, this option allows the software to define the default rule-based bracket symbol. Clear this option to manually select another bracket symbol that is in the applicable set defined in the rules. This option is the same as the **Rule Based** option for the symbol question on the **Selection** tab of the **Bracket System Properties** dialog box. The symbol question is always the final question. For more information, see *Selection Tab (Bracket System Properties Dialog Box)* (on page 267).

NOTES

- When using **Create by Rule** , **Bracket** and **Rule Based** are inactive. You must first click **Select Bracket Supports** , select supports, and click **Accept** to allow the bracket rules to determine the applicable options.

Select Plane

Click to select the type of plane upon which you want to place the bracket system. See *Plane Controls* below.

Select Bracket Supports

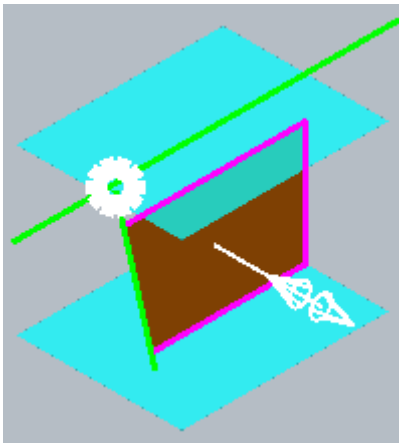
Define the structural support systems for the bracket system. Brackets require at least two, but not more than five, supports. Supports can be plate systems or profile stiffener systems. See *Bracket Support Controls* below.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** / is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Define Boundaries

Define the outside boundaries of the bracket plate system. You can define the boundaries by selecting objects, including structural members, in the model or by sketching boundary curves. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See *Boundary Controls* and *Plane Controls* below.

NOTE When selecting an existing plate system, if  displays for **Define Boundaries**, then the boundaries do not form a closed shape because a boundary has been modified. This condition most commonly occurs when the plate system has a sketched boundary that no longer intersects a plate system or reference plane boundary. Click **Define Boundaries**  to display the location of the gap in the boundary.



 **Select Bracket Reinforcement**

Adds reinforcement to the bracket. See *Bracket Reinforcement Controls* below.

Finish

Places the system using the defined parameters.

 **Cancel**

Clears all selected planes.

 **Accept**

Accepts all selected planes and supports, runs the bracket rules, and determines the catalog symbols that appear in the **Bracket** box.

 **Auto Finish Mode**

Click to set this option to **On** and automatically place a bracket in the model as you define all of the required parameters. When this option is set to **Off**, you must click **Finish** to place the bracket. When this option is set to **On**, some parameters for two-support bracket or three-support bracket are retained for creation of the next bracket.

- For a two-support bracket, if you select a plate as the first support, then that plate is retained as Support 1. You only need to select a second support for subsequent brackets. When creating a sketched two-support bracket, if Support 1 and Support 2 are profiles, then the sketched contour is retained for subsequent brackets. You can select new supports without editing the contour in Sketch 2D.
- For a three-support bracket, if you select a plate as the first support and the third support penetrates Support 1, then only Support 1 is retained for the next bracket. If Support 3 does not penetrate Support 1, then Support 1 and Support 3 are retained. You only need to select Support 2 for subsequent brackets.

NOTE To prevent the software from retaining the supports for subsequent brackets, click **Auto Finish Mode** again to set it to **Off** without exiting the **Bracket Plate System** command.

Connection

Select the type of connection between the bracket and supports: **By Rule**, **Trimmed**, **Lapped**, or **By Reference**. The selected connection is used to guide bracket selection in the rules. This option is not available if a bracket symbol is manually selected in **Bracket Symbol**.

Override UV

Select to override the U and V offsets for all three points that are used to define the plane of a tripping bracket and set the offsets to zero. For more information, see *Tripping Bracket Connect Points Data Tab (Bracket Plate System Properties Dialog Box)* (on page 1447).

Constraint

Select the orientation of a tripping bracket (or bracket by support) with its support. Select **Parallel**, **Perpendicular**, **Transverse**, **Vertical**, **Longitudinal**, **Fixed X**, **Fixed Y**, or **Fixed Z**.

This option is only available when **Plane by Selected Supports** is selected for **Select Plane**.

Orientation

Select the orientation of the bracket: right side, left side, or both sides. This option is only available for a one-support bracket on a profile.

Toggle Dir

Click to move bracket placement to another location using the same combination of supports. Each click toggles the location to the next acceptable location.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

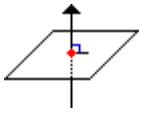
Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Plane Controls

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Plane by Selected Supports

Places the bracket plate system on a plane defined by its supports, such as a plate support along one edge, and a profile or plate support along the other edges. See *Bracket Support Controls* below. A *tripping bracket* is commonly placed by this method and is supported on the flange side of one or more profiles. For more information, see *Bracket by Supports* (on page 247). This is the default plane option for **Select Shape from Standard Catalog Browser** .

Bracket Support Controls

Support 1

Select the first support for the bracket system.

Support 2

Select the second support for the bracket system.

Support 3

Select the third support for the bracket system (For 3S, 4S, and 5S brackets).

Support 4

Select the fourth support for the bracket system (For 4S and 5S brackets).

Support 5

Select the fifth support for the bracket system (For 5S brackets).

Length 1

Type a value to define a point along **Support 1** on the bracket plane.

- If the support is linear, then the **Length 1** box does not display.
- If the support is curved, then the **Length 1** value can be adjusted to change the bracket plane position relative to the support.

Length 2

Type a value to define a second point along **Support 2** on the bracket plane.

- If the support is linear, then the **Length 2** box does not display.
- If the support is curved, then the **Length 2** value can be adjusted to change the bracket plane position relative to the support.

NOTES

- **Length 1** and **Length 2** values affect the position of the bracket plane. These values do not affect bracket creation.
- The software cannot create the bracket If a bracket dimension is greater than a support dimension.

Boundary Controls



Pick Boundaries

Selects objects in the model or in the Workspace Explorer to define the boundaries. This option is only available when you are defining the boundaries.

TIP After you select a boundary plate, press SHIFT to select the **Pick Edges** option. Press SHIFT again to return to the **Pick Boundaries** option.



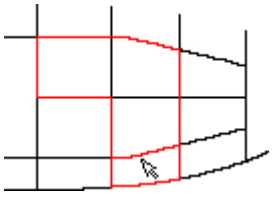
Pick Edges

Selects free edges on plates to define the boundary. This option is available when you select a plate in the **Pick Boundaries** step.



Solve Ambiguity

Clarifies the required boundary. If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to specify the boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).





Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).



Boundary Curve

Draws the boundaries. This option is only available when you are defining the boundaries. See *Define Plane Controls* below.



Sketching Plane

Selects the sketching plane for the boundary.



Add Intersecting Item

Allows you to select objects in the model that intersect the sketching plane to use as reference geometry. You see the selected objects in the Sketch 2D environment.



Add Projection Item

Allows you to select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment.



Sketch 2D

Opens the Sketch 2D environment in which you can draw the boundary.

Auto

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. The Workspace Explorer and the graphics view highlight items in the select set. If the software adds no objects to the select set, a message displays in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Bracket Reinforcement Controls

By Rule

Select to limit reinforcement selection to the default defined in the rules.

Reinforcement

If **By Rule** is cleared, select the reinforcement type: **None**, **Buckling Stiffener**, **Edge Reinforcement**, or **Flange**.

The following options are only available when you select **Edge Reinforcement** or **Buckling Stiffener** for **Reinforcement** and after you make a reinforcement selection and click **Accept**:

Bracket Reinforcement Properties

Click to open the **Bracket Reinforcement Properties** dialog box, which you use to view and modify the properties of the bracket stiffener or edge reinforcement. The initial bracket properties default from the selected structural specification and the bracket and reinforcement rules. For more information, see *Bracket Reinforcement Properties Dialog Box* (on page 271).

NOTE When **Flange** is selected, properties are displayed on the **Flange Selection** tab of the **Bracket System Properties** dialog box. For more information, see *Flange Selection Tab (Bracket System Properties Dialog Box)* (on page 269).

Override

Select to override the rule-based values for **Section Size**, **Material**, and **Grade** and manually select different values. This option is only available after bracket creation when **Edge Reinforcement** or **Buckling Stiffener** is selected for **Reinforcement**.

Section Size

Displays the rule-based profile cross-section of the bracket reinforcement. Select **Override** to modify this value. This option is only available after bracket creation when **Edge Reinforcement** or **Buckling Stiffener** is selected for **Reinforcement**.

Material

Displays the rule-based material type of the bracket reinforcement. **Override** to modify this value. This option is only available after bracket creation when **Edge Reinforcement** or **Buckling Stiffener** is selected for **Reinforcement**.

Grade

Displays the rule-based material grade of the bracket reinforcement. Select **Override** to modify this value. This option is only available after bracket creation when **Edge**

Reinforcement  or **Buckling Stiffener**  is selected for **Reinforcement**.

Primary Orientation

The primary direction of reinforcement placement, based on the molded conventions of the bracket. For more information, see *Molded Conventions Tab (Bracket System Properties Dialog Box)* (on page 254). This option is only available after bracket creation when **Edge Reinforcement**  or **Buckling Stiffener**  is selected for **Reinforcement**.

General Controls

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Bracket Best Practices (on page 50)

What do you want to do?

- *Create a one support bracket by plane* (on page 237)
- *Create a two support bracket by plane* (on page 238)
- *Create a three support bracket by plane* (on page 239)
- *Create a four or five support bracket by plane* (on page 239)

- *Create a two support bracket by supports* (on page 241)
- *Create a three, four, or five support bracket by supports* (on page 241)
- *Manually select a bracket symbol* (on page 242)
- *Sketch a bracket* (on page 243)
- *Add reinforcement to a bracket* (on page 244)
- *Create a bracket and bracket reinforcement using the icon browser* (on page 244)
- *Modify a bracket* (on page 245)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a bracket* (on page 245)

Create a one support bracket by plane

This workflow creates one or two brackets on the web and flange of a profile system.

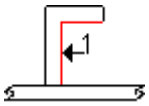


1. Click **Bracket Plate System** on the vertical toolbar.
2. On the **Bracket Plate System** ribbon, click ▼ and select one of the following options to define the plane for the bracket using one of the following methods:

   Coincident Plane - *Define a coincident plane* (on page 76)
   Offset from a Plane - *Define an offset plane* (on page 77)
   Plane by Point and Vector - *Define a plane using a point and a normal vector* (on page 78)
   Plane by Three Points - *Define a plane using three points* (on page 79)

   ★IMPORTANT The plane must intersect the bracket support.
3. Click **Select Bracket Supports**.

   *The ribbon advances to **Support 1**.*
4. In a graphic view or the **Workspace Explorer**, select the profile system that you want to use.



5. If the profile has a flange on more than one side, select an option for **Orientation** to select the side for a single bracket, or to place a bracket on each side.
6. Click **Accept**.

   *A rule-based value for **Bracket** appears in the ribbon and a preview of the bracket appears in the graphic view.*
7. If needed, select a different rule-based value for **Bracket**.
8. If you want to override the rules and manually select a bracket, see *Manually select a bracket symbol* (on page 242).
9. In the **Parent System** box, select a parent system for the bracket.

10. If needed, click **Properties** to further refine attributes, defaults, selection results, or parameters for the bracket.
11. To add a buckling stiffener, edge reinforcement, or flange to the bracket, see *Add reinforcement to a bracket* (on page 244).
12. Click **Finish**.

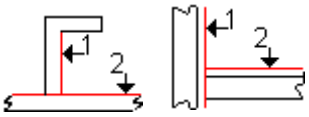
Create a two support bracket by plane

1. Click **Bracket Plate System** on the vertical toolbar.
2. On the **Bracket Plate System** ribbon, click ▼ and select one of the following options to define the plane for the bracket using one of the following methods:

   Coincident Plane - *Define a coincident plane* (on page 76)
   Offset from a Plane - *Define an offset plane* (on page 77)
   Plane by Point and Vector - *Define a plane using a point and a normal vector* (on page 78)
   Plane by Three Points - *Define a plane using three points* (on page 79)

   ★IMPORTANT The plane must intersect the bracket supports.
3. Click **Select Bracket Supports**.

   *The ribbon advances to **Support 1**.*
4. In a graphic view or the **Workspace Explorer**, select the first plate, profile, or member system support.



5. Click **Support 2** and select the second plate, profile, or member system support.
6. Click **Accept**.

   *A rule-based value for **Bracket** appears in the ribbon and a preview of the bracket appears in the graphic view.*
7. If needed, click **Toggle Dir** to change the location of the bracket.
8. If needed, select a different rule-based value for **Bracket**.
9. If you want to override the rules and manually select a bracket, see *Manually select a bracket symbol* (on page 242).
10. In the **Parent System** box, select a parent system for the bracket.
11. If needed, click **Properties** to further refine attributes, defaults, selection results, or parameters for the bracket.
12. To add a buckling stiffener, edge reinforcement, or flange to the bracket, see *Add reinforcement to a bracket* (on page 244).
13. Click **Finish**.

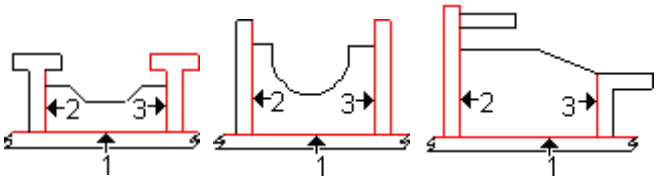
Create a three support bracket by plane

1. Click **Bracket Plate System** on the vertical toolbar.
2. On the **Bracket Plate System** ribbon, click ▼ and select one of the following options to define the plane for the bracket using one of the following methods:

   Coincident Plane - *Define a coincident plane* (on page 76)
   Offset from a Plane - *Define an offset plane* (on page 77)
   Plane by Point and Vector - *Define a plane using a point and a normal vector* (on page 78)
   Plane by Three Points - *Define a plane using three points* (on page 79)

   ★IMPORTANT The plane must intersect the bracket supports.
3. Click **Select Bracket Supports**.

   *The ribbon advances to **Support 1**.*
4. In a graphic view or the **Workspace Explorer**, select the first plate, profile, or member system support.



5. Click **Support 2** and select the second plate, profile, or member system support.
6. Click **Support 3** and select the third plate, profile, or member system support.
7. Click **Accept**.

   *A rule-based value for **Bracket** appears in the ribbon and a preview of the bracket appears in the graphic view.*
8. If needed, click **Toggle Dir** to change the location of the bracket.
9. If needed, select a different rule-based value for **Bracket**.
10. If you want to override the rules and manually select a bracket, see *Manually select a bracket symbol* (on page 242).
11. In the **Parent System** box, select a parent system for the bracket.
12. If needed, click **Properties** to further refine attributes, defaults, selection results, or parameters for the bracket.
13. To add a buckling stiffener, edge reinforcement, or flange to the bracket, see *Add reinforcement to a bracket* (on page 244).
14. Click **Finish**.

Create a four or five support bracket by plane

1. On the **Bracket Plate System** ribbon, click ▼ and select one of the following options to define the plane for the bracket using one of the following methods:

Coincident Plane - *Define a coincident plane* (on page 76)
Offset from a Plane - *Define an offset plane* (on page 77)
Plane by Point and Vector - *Define a plane using a point and a normal vector* (on page 78)
Plane by Three Points - *Define a plane using three points* (on page 79)

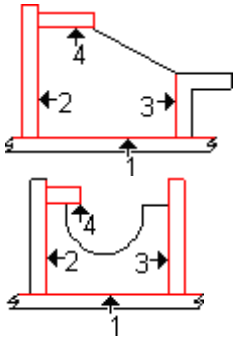
IMPORTANT The plane must intersect the bracket supports.

2. Click **Select Bracket Supports**.

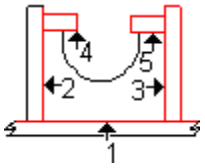
   *The ribbon advances to **Support 1**.*

3. In a graphic view or the **Workspace Explorer**, select the first plate, profile, or member system that you want to use.

   For a four support bracket:

   For a five support bracket:

4. Click **Support 2** and select the second plate, profile, or member system support.
5. Click **Support 3** and select the third plate, profile, or member system support.
6. Click **Support 4** and select the fourth plate, profile, or member system support.
7. For a five support bracket only, click **Support 5** and select the fifth plate, profile, or member system support.
8. Click **Accept**.

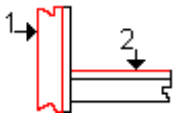
   *A rule-based value for **Bracket** appears in the ribbon and a preview of the bracket appears in the graphic view.*

9. If needed, select a different rule-based value for **Bracket**.
10. If you want to override the rules and manually select a bracket, see *Manually select a bracket symbol* (on page 242).
11. In the **Parent System** box, select a parent system for the bracket.
12. If needed, click **Properties** to further refine attributes, defaults, selection results, or parameters for the bracket.
13. To add a buckling stiffener, edge reinforcement, or flange to the bracket, see *Add reinforcement to a bracket* (on page 244).
14. Click **Finish**.

Create a two support bracket by supports

This workflow creates a bracket oriented by its supports. One or more of the supports may be profile systems.

1. Click **Bracket Plate System** on the vertical toolbar.
2. On the ribbon, click **Plane by Selected Supports** as the plane option.
3. Select the first plate or profile system support.



4. Select the second plate or profile system support.
5. Click **Accept**.

   *A rule-based value for **Bracket** appears in the ribbon and a preview of the bracket appears in the graphic view.*

6. If needed, click **Toggle Dir** to change the location of the bracket.
7. If needed, select a new orientation value for **Constraint**.
8. If needed, select a different rule-based value for **Bracket**.
9. If you want to override the rules and manually select a bracket, see *Manually select a bracket symbol* (on page 242).
10. In the **Parent System** box, select a parent system for the bracket.
11. If needed, click **Properties** to further refine attributes, defaults, selection results, or parameters for the bracket.
12. To add a buckling stiffener, edge reinforcement, or flange to the bracket, see *Add reinforcement to a bracket* (on page 244).
13. Click **Finish**.

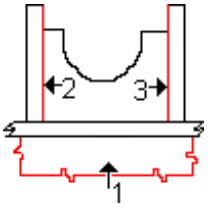
TIP Select **Auto Finish Mode** to place brackets without clicking **Finish** and to retain the first support when placing consecutive brackets.

Create a three, four, or five support bracket by supports

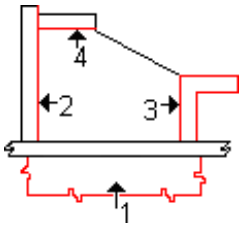
This workflow creates a bracket oriented by its supports. One or more of the supports may be profile systems.

1. Click **Bracket Plate System** on the vertical toolbar.
2. On the ribbon, click **Plane by Selected Supports** as the plane option.
3. Select the first plate or profile system support.
4. Select the third, fourth, and fifth plate or profile system supports as needed.

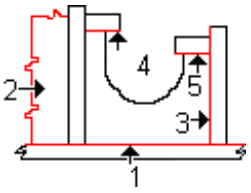
For a three support bracket:



For a four support bracket:



For a five support bracket:



5. Click **Accept** .

   *A rule-based value for **Bracket** appears in the ribbon and a preview of the bracket appears in the graphic view.*

6. If needed, select a new orientation value for **Constraint**.
7. If needed, select a different rule-based value for **Bracket**.
8. If you want to override the rules and manually select a bracket, see *Manually select a bracket symbol* (on page 242).
9. In the **Parent System** box, select a parent system for the bracket.
10. If needed, click **Properties** to further refine attributes, defaults, selection results, or parameters for the bracket.
11. To add a buckling stiffener, edge reinforcement, or flange to the bracket, see *Add reinforcement to a bracket* (on page 244).
12. Click **Finish**.

TIP For three-support brackets, select **Auto Finish Mode** to place brackets without clicking **Finish** and to retain the first and third supports when placing consecutive brackets.

Manually select a bracket symbol

To override the rules and manually select a bracket when using **Plane by Selected Supports** , perform the following steps:

1. If **Create by Rule** is visible, then click the button.

   *The button toggles to **Select from the Catalog** , and **Rule Based** is hidden.*

2. Select **More** in the **Bracket** box.

   *The **Catalog Objects** dialog box appears.*

3. Select a bracket symbol, and click **OK**
4. Click **Select Bracket Supports** and continue with bracket creation.
5. To add a buckling stiffener, edge reinforcement, or flange to the bracket, see *Add reinforcement to a bracket* (on page 244).
6. Click **Finish**.

Sketch a bracket

If you want to sketch the free edges of a bracket, perform the following steps:

1. Click **Bracket Plate System** on the vertical toolbar.
2. On the **Bracket Plate System** ribbon, click , and select the needed plane or support method.
3. If **Create by Rule** is visible, then click the button.

   *The button toggles to **Select from the Catalog**, and **Rule Based** is hidden.*
4. Select **More** in the **Bracket** box**.**

   *The **Catalog Objects** dialog box appears.*
5. Select a bracket symbol that supports sketching, and click **OK**.

   NOTE Delivered symbols that support sketching have *Sketched* in their names.
6. Click **Select Bracket Supports**, and select the needed bracket supports.
7. Click **Define Boundaries**.
8. Click **Boundary Curve**.
9. Click **Add Intersecting Item**.
10. Select the bracket supports and other objects in the model that intersect the bracket sketching plane. These extra objects appear in the Sketch 2D environment as reference geometry.
11. Click **Add Projection Item**.
12. Select objects in the model that do not intersect the sketching plane that you want to use when sketching the bracket. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment as reference geometry.
13. Click **Sketch 2D**.
14. In the Sketch 2D environment, sketch the bracket.

NOTE You can constrain the bracket to geometric construction points referenced from profile cross-section key points or offset from plate systems. For more information, see *Point at Offset from Keypoint Command* and *Point at Minimum Distance Command* in the *SmartSketch Drawing Editor Drawing Editor Help*, available within the Sketch 2D environment.

1. Click **Finish** in Sketch 2D.
2. In the **Parent System** box, select a parent system for the bracket.
3. If needed, click **Properties** to further refine attributes, defaults, selection results, or parameters for the bracket.
4. Click **Finish**.

Add reinforcement to a bracket

1. Do one of the following:
   - In a graphic view or the **Workspace Explorer**, select an existing bracket system.
   - When creating a new bracket, click **Select Bracket Supports**, and select the supports.

   *The **Bracket Plate System** ribbon appears.*
2. If **Create by Rule** is visible, then click the button.

   *The button toggles to **Select from the Catalog**, and **Rule Based** is hidden.*
3. Click **Select Bracket Reinforcement**.
4. Select a value for **Reinforcement** to select a buckling stiffener, edge reinforcement, or flange.
5. Click **Accept**.

   Flange parameters appear in the ribbon and a preview of the bracket appears in the graphic view.
6. Change the flange parameters as needed.
7. Click **Finish**.

NOTE To automatically place the default reinforcement defined in the catalog rules, select **Rule Based** when **Filter by Rules** is visible.

Create a bracket and bracket reinforcement using the icon browser

This workflow selects a bracket and bracket reinforcement in a single dialog box before selecting the supports. The dialog box can remain open to simplify creation of successive brackets.

1. Click **Bracket Plate System** on the vertical toolbar.
2. On the **Bracket Plate System** ribbon, click ▼, and select one of the following options to define the plane for the bracket using one of the following methods:

   Coincident Plane - *Define a coincident plane* (on page 76)
   Offset from a Plane - *Define an offset plane* (on page 77)
   Plane by Point and Vector - *Define a plane using a point and a normal vector* (on page 78)
   Plane by Three Points - *Define a plane using three points* (on page 79)

   Plane by Selected Supports - Select this option to create a tripping bracket.

   ★IMPORTANT The plane must intersect the bracket support.
3. Click **Bracket Browser**.

   *The **Define Bracket Plate System Properties** dialog box appears.*
4. To manually select a bracket, clear **By Rule**.

5. To manually select a bracket reinforcement, clear **Reinforcement By Rule**.
6. Select a bracket and reinforcement type.
7. Set bracket properties and parameters.
8. Click **Apply**.
9. With the dialog box open, click **Select Bracket Supports** on the ribbon.
10. In a graphic view or the **Workspace Explorer**, select the profile system that you want to use.
11. Select supports, connection type, and other parameters on the ribbon.
12. Click **Finish**.

    The bracket is created.

Create additional brackets using the icon browser

1. To create another bracket of the same plane type, click the **Define Bracket Plate System Properties** dialog box to make it the active window and repeat the previous steps.
2. To create a bracket of a different plane type, click **Select Plane** on the ribbon, and then select a different option. Click the **Define Bracket Plate System Properties** dialog box to make it the active window, and repeat the previous steps.
3. Click **Close** to close the **Define Bracket Plate System Properties** dialog box.

Modify a bracket

1. Click **Select** on the vertical toolbar.
2. Select **Root Plates** in the **Locate Filter**.
3. Select one or more brackets to modify in a graphic view or in the **Workspace Explorer**.
4. Using the ribbon controls, edit the brackets as needed.

NOTES

- If you select multiple brackets, you can edit common properties on the ribbon.
- If you select multiple brackets by supports, you can modify load points and offsets for the brackets. For more information, see *Bracket Connect Points Data Tab (Bracket System Properties Dialog Box)* (on page 271).
- If the bracket is constrained to geometric construction points, you can modify the points. A separate ribbon displays for the points. For more information, see *Points (Geometric Construction Palette Dialog Box)* (on page 1028).

Delete a bracket

1. Click **Select** on the vertical toolbar.
2. Select **Bracket Plate Systems** in the **Locate Filter**.
3. Select the bracket system to delete.
4. Click **Delete**.

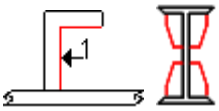
Bracket by Plane

A bracket system can have a plane that intersects the supports. This type of bracket is created when **Coincident Plane**, **Offset from a Plane**, **Angle from plane**, **Plane by Point and Vector**, **Plane by Three Points**, or **Plane by Vectors Normal** are selected for **Select Plane**.

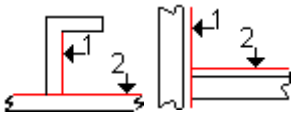
The following examples show the general support selection sequence:

1 Support (1S) Bracket

Typically a profile with the bracket on the flange and web.



2 Support (2S) Bracket

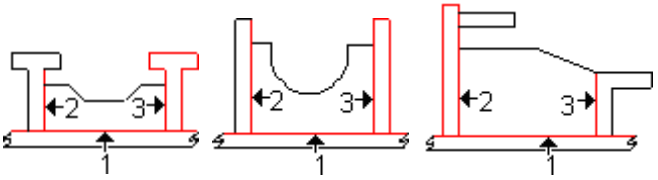


Examples of supports include:

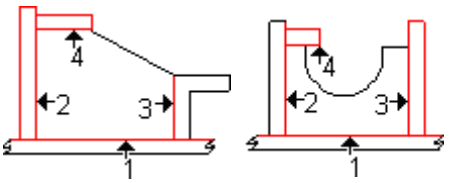
- Two plate systems
- A plate system and an edge reinforcement
- A rib connection, using a plate system and the web of a profile



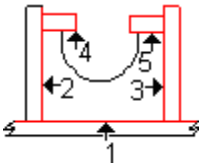
3 Support (3S) Bracket



4 Support (4S) Bracket



5 Support (5S) Bracket

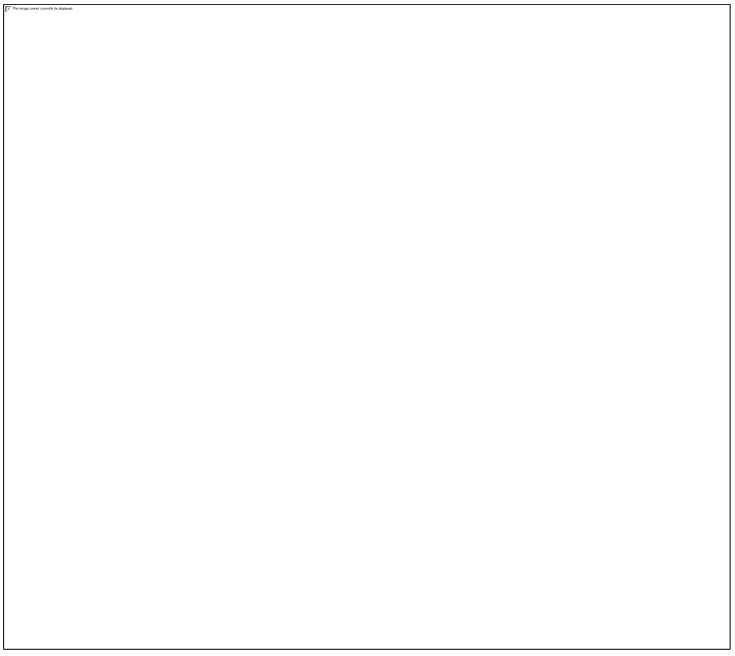


Bracket by Supports

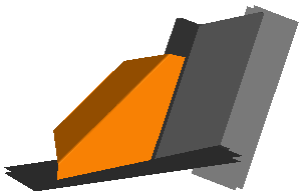
A bracket system can have a plane that is in line with one of its selected supports. This type of bracket is created when **Plane by Selected Supports** is selected for **Select Plane**. You can select a bracket symbol or sketch the free edge of the bracket.

The supports are used to define the plane of the bracket as well as its boundaries. The first two supports must intersect in a manner that allows the plane to be defined. Bracket orientation is driven by the orientation of one of the supports, such as a plate system or the web of a profile system.

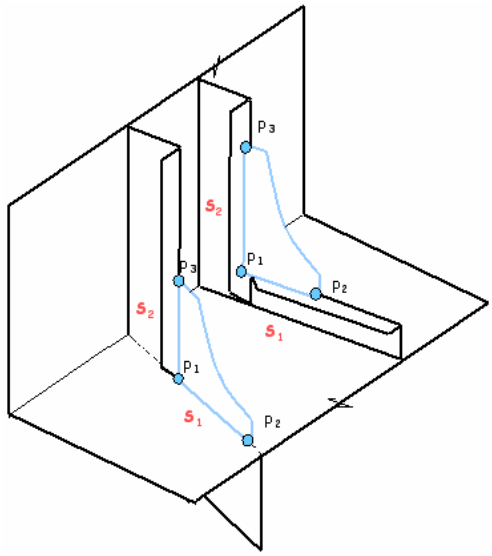
NOTE The default bracket orientation is parallel to the first selected support having a plane usable to drive the orientation.



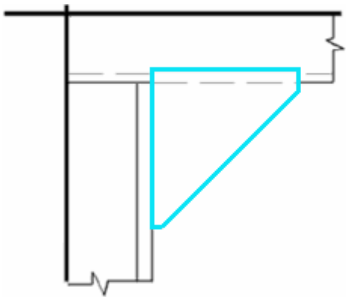
A flange can be added to the free edge of the bracket.



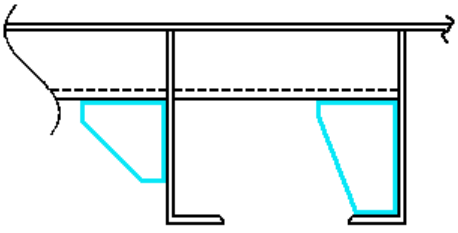
A bracket can be supported on the flange side of one or more profiles. It is created in the same way as other brackets except that the plane of the bracket is usually offset so that it can be fillet welded to the supporting profile flange.



A bracket by supports can be lapped to one or more supports.

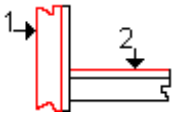


A bracket by supports can use two intersecting profiles on the same plate as supports. The bracket can be supported by the web, or both the web and profile of the second profile.

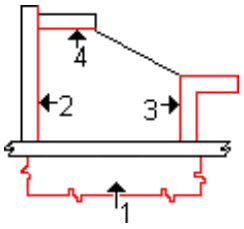


A bracket is classified by its number of supports. The support object type selected affects the available support object types available for succeeding supports. The following examples show the general support selection sequence and the available combinations of supports:

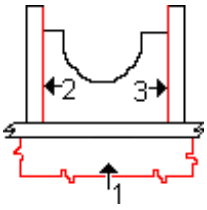
2 Support (2S) Bracket



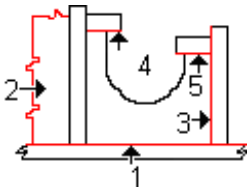
4 Support (4S) Bracket



3 Support (3S) Bracket



5 Support (5S) Bracket



Support 1 Object	Allowed Support 2 & 3 Objects	Allowed Support 4 & 5 Objects
Plate edge	Plate face Plate edge Profile system	Plate face Profile system
Plate face	Plate edge Profile system	Plate face Profile system
Profile system	Plate face Plate edge Profile system	Plate face Profile system

Support Order when Supports are Unconnected

Bracket supports are usually connected to each other, either by bounding or intersecting. There are some circumstances where supports are not connected. The supports must be linear (for profiles) and planar (for plates) so that the intersection of the unbounded landing curves and/or unbounded definition planes are unique.

Bracket System Properties Dialog Box

Specifies the properties for the root or leaf bracket system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Bracket System Properties Dialog Box) (on page 250)

Material Tab (Bracket System Properties Dialog Box) (on page 253)

Molded Conventions Tab (Bracket System Properties Dialog Box) (on page 254)

Selection Tab (Bracket System Properties Dialog Box) (on page 267)

Parameters Tab (Bracket System Properties Dialog Box) (on page 268)

Flange Selection Tab (Bracket System Properties Dialog Box) (on page 269)

Flange Parameters Tab (Bracket System Properties Dialog Box) (on page 270)

Bracket Connect Points Data Tab (Bracket System Properties Dialog Box) (on page 271)

Main Tab (Bracket System Properties Dialog Box)

Specifies the general properties of the bracket plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

Specifies the level of tightness as it applies to the entire plate system.

Active

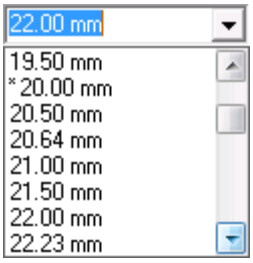
Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on

page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Edge Length Thickness Ratio (L/T) - Displays the length divided by the thickness. This value is used by the rules to calculate the reinforcement.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Bracket System Properties Dialog Box)

Specifies the material properties for the bracket that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

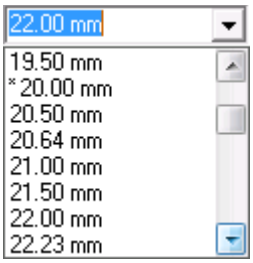
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Bracket System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the bracket system.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



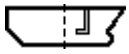
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



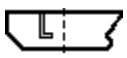
Aft - Orients profile flanges aft.



Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



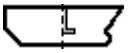
Aft - Orients profile flanges aft.



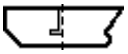
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



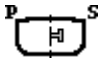
Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



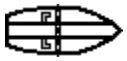
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



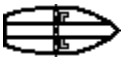
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



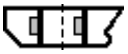
Outboard - Positions thickness away from the forward/aft reference.



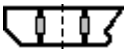
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

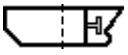
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



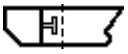
Aft - Orients profiles aft.



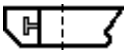
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



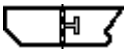
Aft - Orients profiles aft.



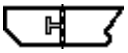
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



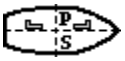
Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

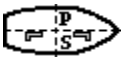
Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



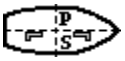
Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



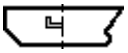
Starboard - Orients profile flanges to the starboard side.



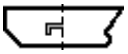
Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



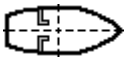
Aft - Orients profile flanges aft.



Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



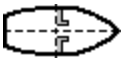
Aft - Orients profile flanges aft.



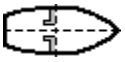
Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

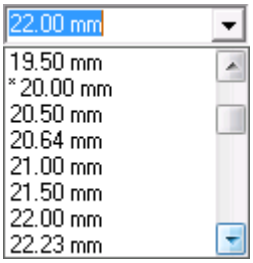


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Selection Tab (Bracket System Properties Dialog Box)

Specifies the rule criteria used for bracket plate system selection.

TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.
- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or

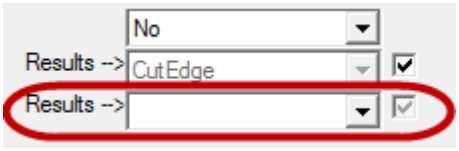
answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.
- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.



1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.
2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

Results - Select the bracket class.

Bracket Contour Type - Select a bracket type. Select the check box to use the default answer of the rule class.

Results - Select the bracket catalog item. Displays only the valid selections based on the **Bracket Contour Type** answer.

IMPORTANT The questions described here represent the default rules delivered with the software. Customized rules may have different questions.

Parameters Tab (Bracket System Properties Dialog Box)

Specifies the parameters for the bracket plate system.

Parameter - Displays the name of the parameter.

Value - Displays the value of the parameter. Values that are shaded cannot be modified.

Rule-Based - Displays a check box if the parameter value is rule-based. Clear the check box to modify the value in the **Value** box.

Catalog Value - Displays the original catalog value if the **Value** box has been modified.

Flange Selection Tab (Bracket System Properties Dialog Box)

Specifies the rule criteria used for bracket flange selection. This tab is only visible after bracket creation when **Flange** is selected for **Reinforcement**.

TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.
- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.
- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.



1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.
2. Click the **Rule Based** box again to select and set all objects to rule based.

User Answers

Select from a list of bracket flanges. The default answers are determined by the rule class.

Results - Select the bracket flange class.

IMPORTANT The questions described here represent the default rules delivered with the software. Customized rules may have different questions.

Flange Parameters Tab (Bracket System Properties Dialog Box)

Specifies the default values for the bracket flange parameters. Values can be modified as permitted by the catalog. This tab is only visible after bracket creation when **Flange** is selected for **Reinforcement**.

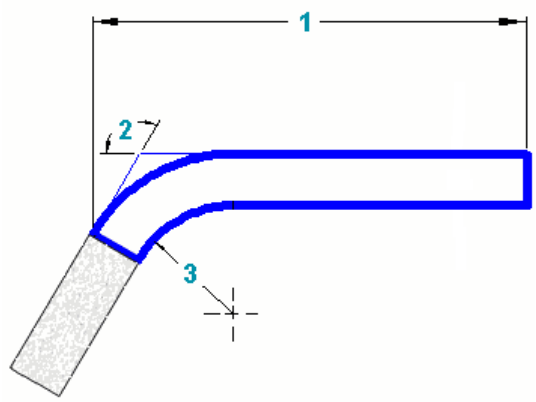
Neutral Axis Ratio - Sets the molded side of the bracket and bracket flange. This option creates the flange symbol at the correct location by accounting for molded convention thickness direction and offset values.

- If the molded side is the inner surface, the value is 0.
- If the molded side is the outer surface, the value is 1.
- If the molded side is centered, the value is 0.5.

Flange Width - Width of the flange measured from the edge of the bracket symbol, which is also the first bend tangent.

Inner Radius - Bend radius measured along the inner surface of the bend.

Bend Angle - Angle between the flange surface and bracket surface.



1 Flange Width

2 Bend Angle

3 Inner Radius

KnuLine - The knuckle line for the flange, measured from the edge where the flange is added to the bracket plate to the edge of the unrolled flange. This is calculated by the rule based on the other parameter values so that when the flange is fully bent at manufacturing, the flange length is correct as specified. This parameter is read-only.

RollLine1 / RollLine2 - The roll lines define exactly how the flange is rolled or bent during manufacturing. **RollLine1** is measured from the edge where the flange is added to the bracket plate, and **RollLine2** is measured from where **RollLine1** ends. These parameters are read-only and are calculated by the rule based on the other parameter values.

Flange Orientation -Select the orientation of the flange bracket.

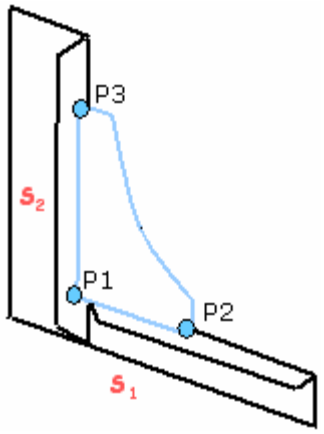
- **Same As Parent** - Orientation is the same as the bracket plate system primary profile orientation.
- **Against Parent** - Orientation is in the opposite direction of the bracket plate system primary profile orientation.

Bracket Connect Points Data Tab (Bracket System Properties Dialog Box)

Specifies the default offset values and lapped status for a bracket by supports. This tab is only visible after bracket creation when **Plane by Selected Supports** is selected for **Select Plane**.

Offset Values

Load Point - Specifies the support load point of each point used to calculate the bracket system plane. Offsets are measured from the load points. Load points are defined in the profile cross-section symbols. If one of the supports is a plate system, then the load point value is 0. For more information, see *Key Points* (on page 1353).



U Offset - Specifies the offset along the local U axis of the support profile. The value is 0 if the bracket is not on the flange of a profile support.

V Offset - Specifies the offset along the local V axis of the support profile. The value is 0 if the bracket is not on the flange of a profile support.

Override - Select to specify different values for **Load Point**, **U Offset** and **V Offset**.

Lapped Status

Displays the status of each bracket support. **Lapped** is selected if the support is lapped. This value is set during bracket creation and cannot be changed here.

Bracket Reinforcement Properties Dialog Box

Specifies the properties for the bracket system reinforcement that you are editing. Tabs on this dialog box are visible only after bracket creation.

Cross Section Tab (Bracket Reinforcement Properties Dialog Box) (on page 272)
Orientation Rule Tab (Bracket Reinforcement Properties Dialog Box) (on page 272)

Cross Section Tab (Bracket Reinforcement Properties Dialog Box)

Specifies the cross-section settings for a bucking stiffener bracket reinforcement. This tab is only visible after bracket creation when **Edge Reinforcement** or **Buckling Stiffener** is selected for **Reinforcement**.

Section Type

Specifies the profile cross-section type. The software displays the section type properties for the section type that you selected. Each section type has different properties.

Section Size

Specifies the profile section size.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Override Cross Section Information

Select to override the cross-section information generated by the rule.

WebLength(WL)

Displays the web length of the reinforcement.

WebThickness(WT)

Displays the web thickness of the reinforcement.

Orientation Rule Tab (Bracket Reinforcement Properties Dialog Box)

Specifies the orientation settings for a bracket edge reinforcement. This tab is only visible after bracket creation when **Edge Reinforcement** is selected for **Reinforcement**.

Rule

Select the orientation rule that you want to use for the edge reinforcement. Available rules include **Normal**, **Longitudinal**, **Vertical**, and **Transverse**.

Define Bracket Plate System Properties Dialog Box

Specifies the bracket symbol to use prior to choosing supports. To select the bracket symbol and edit its properties, click **Define Bracket Plate System properties** dialog box.

By Rule - Select to limit bracket selection to the default defined in the catalog rules. When cleared, you can manually select a bracket.

The following options are available when you clear **By Rule**.

- **Filters** - Specifies the bracket type. Select **All, 1S, 2S, 3S, 4S, 5S, Sketched**, or **Flanged**.

- *Bracket class tabs* - Each tab displays a bracket class name. The tabs are based on the selected bracket type in **Filter**.
- *Bracket icons* - Each tab displays small icons of individual brackets within the bracket class. Select an icon to display a preview of the bracket.
- **Clear Recent** - Clears the recently selected brackets from **Recent** tab.

Toggle Dir - Click to move bracket placement to another location using the same combination of supports. Each click toggles the location to the next acceptable location.

Reinforcement By Rule - Select to limit bracket reinforcement selection to the default defined in the catalog rules. If **By Rule** is unchecked, select the reinforcement type: **None**, **Buckling Stiffener**, **Edge Reinforcement**, or **Flange**.

Bracket Properties

Bracket tab

Specifies properties for the bracket.

Thickness - Select the material thickness for the plate system.

Grade - Select the material grade for the plate system.

Type - Select the type of plate system that you are placing. If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to Deck. Plate systems that are mostly transverse (YZ plane) are assigned to Transverse Bulkhead. Plate systems that are mostly longitudinal (XZ plane) are assigned to Longitudinal Bulkhead. This option is also used by the naming rule to name the plate system.

Thickness Direction - Select the direction in which to apply the thickness. If you have selected supports for the bracket, the software displays an arrow indicating the active direction.

Buckling Stiffener and Edge Reinforcement tabs

Specifies properties for the bracket reinforcement. These tabs are available when **Buckling Stiffener** or **Edge Reinforcement** are selected for **Reinforcement By Rule.**

Override - Select to override the rule-based values for **Section Size**, **Material**, and **Grade** and manually select different values.

Section Type - Select the profile section type. The software displays the section type properties for the section type that you selected. Each section type has different properties.

Section Size - Displays the rule-based profile cross-section of the bracket reinforcement. Select **Override** to modify this value.

Material - Displays the rule-based material type of the bracket reinforcement. **Override** to modify this value.

Grade - Displays the rule-based material grade of the bracket reinforcement. Select **Override** to modify this value.

Primary Orientation - Specifies the primary direction of reinforcement placement, based on the molded conventions of the bracket. For more information, see *Molded Conventions Tab (Bracket System Properties Dialog Box)* (on page 254).

Orientation Rule - Select the orientation rule that you want to use for the edge reinforcement. Available rules include **Normal**, **Longitudinal**, **Vertical**, and **Transverse**.

Flange tab

Specifies properties for the bracket flange. This tab is available only when **Flange** is selected for **Reinforcement By Rule.**

Flange Shape - Select the flange type.

Flange Orientation -Select the orientation of the flange bracket.

- **Same As Parent** - Orientation is the same as the bracket plate system primary profile orientation.
- **Against Parent** - Orientation is in the opposite direction of the bracket plate system primary profile orientation.

Questions, Answers, and Results

The default answers are determined by the rule class.

Results - Select the bracket class.

Bracket Contour Type - Select a bracket type. Select the check box to use the default answer of the rule class.

Results - Select the bracket catalog item. Displays only the valid selections based on the **Bracket Contour Type** answer.

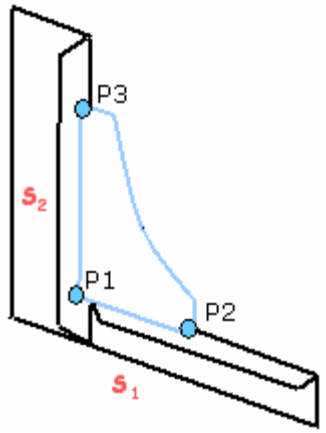
★ **IMPORTANT** The questions described here represent the default rules delivered with the software. Customized rules may have different questions.

Connect Points Frame

Specifies the default offset values for a bracket by supports. This frame is available when **Plane by Selected Supports** is selected for **Select Plane** on the ribbon.

Offset Values

Load Point - Specifies the support load point of each point used to calculate the bracket system plane. Offsets are measured from the load points. Load points are defined in the profile cross-section symbols. If one of the supports is a plate system, then the load point value is 0. For more information, see *Key Points* (on page 1353).



U Offset - Specifies the offset along the local U axis of the support profile. The value is 0 if the bracket is not on the flange of a profile support.

V Offset - Specifies the offset along the local V axis of the support profile. The value is 0 if the bracket is not on the flange of a profile support.

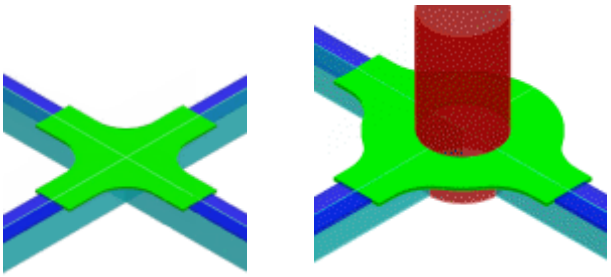
Override - Select to specify different values for **Load Point**, **U Offset** and **V Offset**.

Lapped Status

Displays the status of each bracket support. **Lapped** is selected if the support is lapped. This value is set during bracket creation and cannot be changed here.

Advanced Plate System

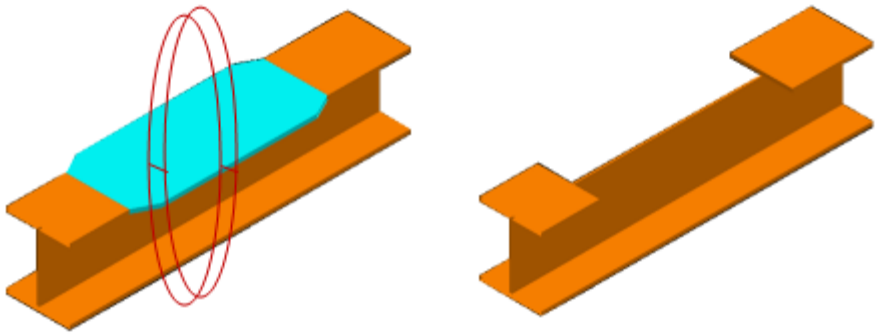
Creates specialized, complex, such as side plates, ring plates, transition plates, continuity plates, and buckling plates.



With this command, you create a predefined plate using an advanced plate system macro or define your own custom plate using geometric construction interfaces.

Using Advanced Plate Systems with Members

Most advanced plate systems are used with standard and built-up members created with **Place Linear Member** in the Molded Forms or Structure tasks. Member flanges and webs are trimmed (rebounded) and connected to the advanced plate system. As an example, when you insert an advanced plate system along the length of a standard (rolled) member, typically on the flange, the software creates a penetration connectionbetween the standard member and the advanced plate system.



When you use **Execute Detailing** to detail the plate, the software automatically creates all connections and penetration seams.

When you modify an existing member, you also must modify connections and trimming by selecting a port and using the **Modify Member Generic Assembly Connection** ribbon in the Molded Forms and Structural Detailing tasks.

For more information, see *Place Linear Member Systems* (on page 787) and *Edit assembly connection ports* (on page 971).

Using Geometric Constructions

When an advanced plate system macro is not available, geometric construction interfaces can be used to create custom plate systems. You can use multiple geometric constructions in sequence. For example, you can create points, create curves from the points, and then create a surface from the curves. Geometric constructions are available by clicking **Plate Definition by 3D** on the ribbon.

Advanced Plate System Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The initial properties default from the selected structural specification and the advanced plate system rules. For more information see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro.
- **APS Icon Browser** - Graphically displays the available advanced plate system macros. For more information, see *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of the available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

NOTE The software displays recently selected macros in the list.

Plate Definition by 3D

Allows the creation of plate geometry directly in a graphic view by using the **Geometric Construction Palette** dialog box, the **Geometric Construction** ribbon, and the **Geometric Construction Explorer**. This option is the 3D equivalent to Sketch 2D and is only available when **Construction** is selected for **Macro Definition**. For more information, see *Geometric Constructions* (on page 1027).

Select Supports

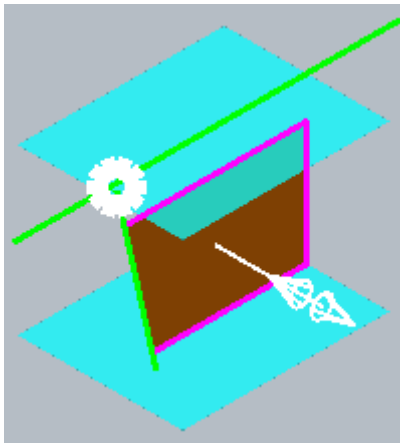
Returns the ribbon to the options available for the selected macro. This option is only available after the initial definition of the advance plate system.

Define Boundaries

Sets the outside boundaries of the plate system. You can define the boundaries by selecting objects, including structural members, in the model or by sketching boundary curves. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See *Define Boundary Controls* below.

NOTE When selecting an existing plate system, if displays for **Define Boundaries**, then the boundaries do not form a closed shape because a boundary has been modified.

This condition most commonly occurs when the plate system has a sketched boundary that no longer intersects a plate system or reference plane boundary. Click **Define Boundaries** to display the location of the gap in the boundary.



Finish

Places the system using the defined parameters.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Define Boundary Controls

Available when you use **Define Boundaries** .

Pick Boundaries

Selects objects in the model or in the Workspace Explorer to define the boundaries. This option is only available when you are defining the boundaries.

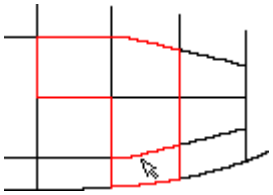
TIP After you select a boundary plate, press SHIFT to select the **Pick Edges** option. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Selects free edges on plates to define the boundary. This option is available when you select a plate in the **Pick Boundaries** step.

Solve Ambiguity

Clarifies the required boundary. If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to specify the boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Draws the boundaries. This option is only available when you are defining the boundaries. See *Define Plane Controls* below.

Sketching Plane

Selects the sketching plane for the boundary.

Add Intersecting Item

Allows you to select objects in the model that intersect the sketching plane to use as reference geometry. You see the selected objects in the Sketch 2D environment.

Add Projection Item

Allows you to select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the boundary.

Auto

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. The Workspace Explorer and the graphics view highlight items in the select set. If the software adds no objects to the select set, a message displays in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

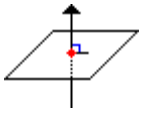
Define Plane Controls

Available when you use **Boundary Curve** .

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

What do you want to do?

- *Place a side plate* (on page 288)
- *Place a side plate with sniped corners* (on page 294)
- *Place an axial side plate* (on page 298)
- *Place a ring plate* (on page 305)

- *Place a ring plate with sniped corners* (on page 312)
- *Place a ring plate with radial edges* (on page 319)
- *Place a transition plate* (on page 332)
- *Place a gap transition plate between flanges* (on page 337)
- *Place a gap transition plate between webs* (on page 339)
- *Place a continuity plate* (on page 325)
- *Place a buckling plate along a built-up boxed member* (on page 344)
- *Place a buckling plate aligned with flanges or continuity plates* (on page 349)
- *Place a buckling plate along a member* (on page 355)
- *Place a buckling plate at a rolled member and a nodal connection* (on page 360)
- *Place a buckling plate along a built-up tubular member* (on page 365)
- *Place a partial buckling plate on one side of a member web* (on page 371)
- *Place a buckling plate on one side of a member web* (on page 375)
- *Place a knuckled bracket* (on page 382)
- *Place a cross insert plate on supporting and supported members* (on page 388)
- *Place a cross insert plate on intersecting members* (on page 392)
- *Place a tee insert plate on supporting and supported members* (on page 396)
- *Place a column insert plate along a member* (on page 400)
- *Place a column insert plate on supporting and supporting members* (on page 405)
- *Modify an advanced plate system* (on page 281)
- *Place a custom plate system using geometric constructions* (on page 282)
- *Modify a plate system created with geometric constructions* (on page 282)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Modify an advanced plate system

1. Click **Select** on the vertical toolbar.
2. Select **Root Plates** in the **Locate Filter**.
3. Select the advanced plate system to modify.

   *The **Advanced Plate System** ribbon displays.*
4. Click **Select Supports**.

   The ribbon for the advanced plate system macro displays.
5. Click **Select** on the ribbon.
6. Change the required inputs and parameters.

7. Click **Continue**.

   *The ribbon display returns to the **Advanced Plate System** ribbon.*

8. Click **Finish**.

Place a custom plate system using geometric constructions

The following workflow is general. The macros to use can vary.

1. Click **Advanced Plate System** on the vertical toolbar.

   *The **Advanced Plate System** ribbon displays.*

2. Select **Construction** from the **Macro Definition** list.

   -OR-

   If **Construction** was selected earlier in the same session, and is visible in the **Macro Definition** list, click **Plate Definition by 3D**.

   *The **Geometric Construction Palette** dialog box, **Geometric Construction Explorer** dialog box, and **Geometric Construction** ribbon display.*

3. If required, select a point geometric construction interface.
4. Follow the system prompts and create the points.
5. Click **Continue**.

   *The **Geometric Construction Explorer** shows a list of points.*

6. Select a curve geometric construction interface.
7. Follow the system prompts and select points to create the needed curves. Select the points from the graphic view, the **Workspace Explorer**, or the **Geometric Construction Explorer**.
8. Click **Continue**.

   *A list of curves is added to the **Geometric Construction Explorer**.*

9. Select a surface geometric construction interface.
10. Follow the system prompts and select curves to create a surface. Select the curves from the graphic view, the **Workspace Explorer**, or the **Geometric Construction Explorer**.
11. Click **Continue**.

    *A surface is added to the **Geometric Construction Explorer**.*

12. Click **Close**.

    *The ribbon display returns to the **Advanced Plate System** ribbon.*

13. Create boundaries for the plate system as required. Boundaries create connections between plates.
14. Click **Finish**.

    The software creates a plate system using the geometric construction surface.

Modify a plate system created with geometric constructions

1. Click **Select** on the vertical toolbar.

2. Select **Root Plates** in the **Locate Filter**.
3. Select the advanced plate system to modify.

   *The **Advanced Plate System** ribbon displays.*
4. Click **Plate Definition by 3D** .

   *The **Geometric Construction Palette** dialog box, **Geometric Construction Explorer** dialog box, and **Geometric Construction** ribbon display.*
5. Click **Select** on the ribbon.
6. Select the geometry on the **Geometric Construction Explorer** or select a numbered input step on the ribbon.

   The geometry highlights in the graphic view.
7. Make the required changes.
8. To delete geometry, click **Delete** .
9. Click **Continue**.

   *The ribbon display returns to the **Advanced Plate System** ribbon.*
10. Click **Finish**.

Advanced Plate Palette Dialog Box

Specifies an advanced plate system macro to use. The dialog box displays when you click **APS Icon Browser** in the **Macro Definition** list. Each tab of the palette graphically displays options available for a type of macro in the Catalog database. After you select a macro, the appropriate options display on the **Advanced Plate System** ribbon.

Filters

Limits the available macros on a tab to the selected type.

Clear Recent

Removes recently selected macros from the **Recent** tab.

Help

Displays help for the selected macro in a preview window. If no help is available, a preview of the selected macro displays.

NOTE You can also select **Tools** > **Automatic Preview** to display the preview window.

Side Plates Tab (Advanced Plate Palette Dialog Box) (on page 284)

Ring Plates Tab (Advanced Plate Palette Dialog Box) (on page 301)

Continuity Plates Tab (Advanced Plate Palette Dialog Box) (on page 321)

Transition Plates Tab (Advanced Plate Palette Dialog Box) (on page 328)

Buckling Plates Tab (Advanced Plate Palette Dialog Box) (on page 341)

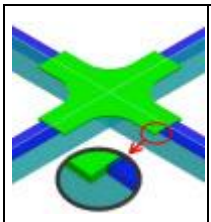
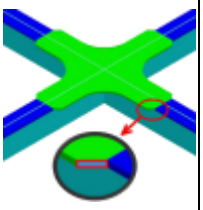
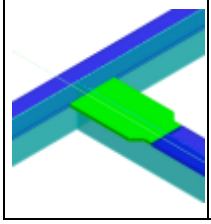
Brackets Tab (Advanced Plate Palette Dialog Box) (on page 378)

Insert Plate Tab (Advanced Plate Palette Dialog Box) (on page 385)

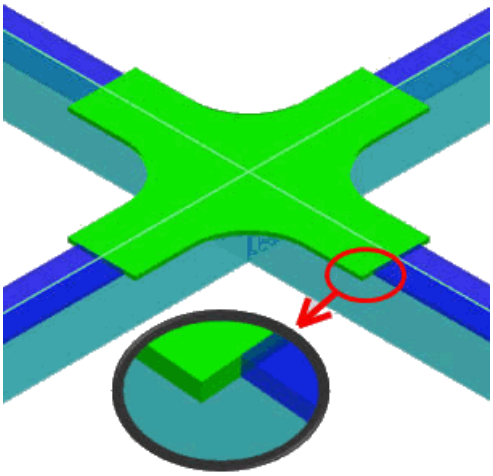
NOTE The **Recent** tab displays macros recently selected on the other tabs. Click **Clear Recent** to remove interfaces from this tab.

Side Plates Tab (Advanced Plate Palette Dialog Box)

The following advanced plate system macros create side plates, which reinforce one or more members sharing a nodal connection. A nodal connection is typically a construction assembly between built-up members and consists of side plates or ring plates, transition plates, and continuity plates.

	**Side Plate** - Creates a side plate by connecting all flanges on a set of incoming members that share a nodal connection. The corners of the side plate are square. For more information, see *Side Plate* (on page 284).
	**Side Plate 2** - Creates a side plate by connecting all flanges on a set of incoming members that share a nodal connection. The corners of the side plate are sniped. For more information, see *Side Plate 2* (on page 290).
	**Side Plate Axial** - Creates a side plate axially on a supported member at its nodal connection with a supporting member. For more information, see *Side Plate Axial* (on page 296).

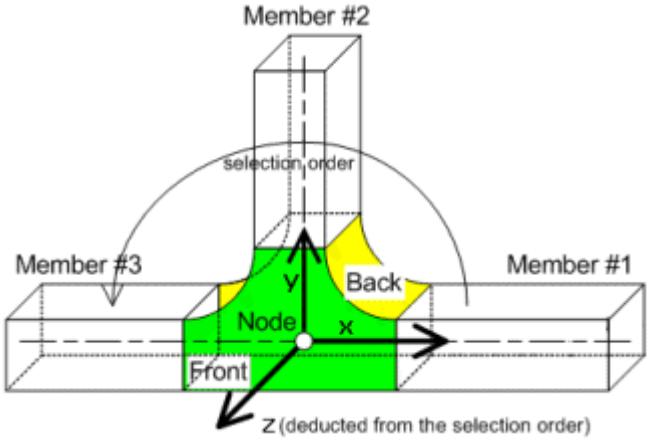
Side Plate



Tooltip: SidePlate

Creates a side plate by connecting all flanges on a set of incoming members that share a nodal connection. The corners of the side plate are square.

The physical axes of the members are end-point connected and coplanar. The common end point is the node. You must have at least two members with non-collinear axes. The incoming flange and web plate parts are trimmed. For built-up members, the software trims theincoming flange and web parts by the edges of the side plate.



The macro computes a local coordinate system from the following:

- X-axis - Defined by the axis of the first selected member oriented from the nodal point to the other end of the member.
- Y-axis - Defined by the axis of the second selected member oriented from the nodal point to the other end of the member.
- Z-axis - Calculated as the cross-product of X-axis by Y-axis.

The type of free edge on the side plate depends on the angle between the axes of two adjacent members.

Side Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).

- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the primary members that support the side plate. These members can be rolled members or built-up members. Their physical axes must be coplanar and connected at the end points. Select the first two members in a clockwise or counter-clockwise direction. The first member that you select defines the X-axis for the side plate. The second member that you select defines the Y-axis for the side plate.

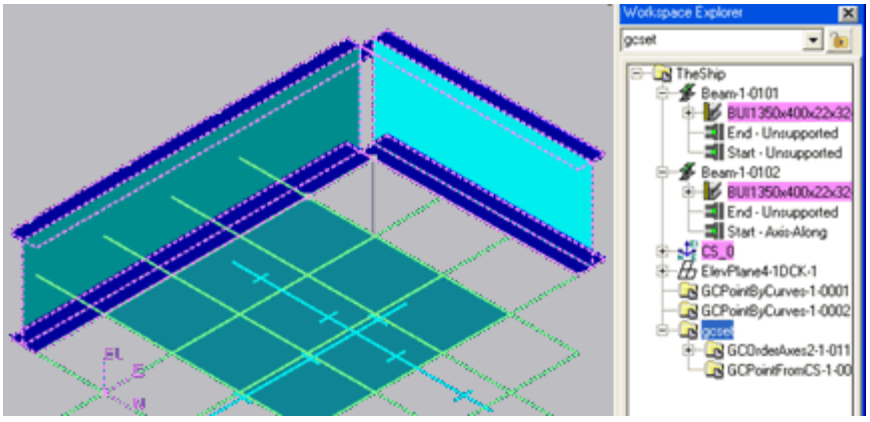
(2) - (Optional) Specifies secondary members that are attached to the face of the side plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

- 1 - No background color when no input is selected.
- 1 - Yellow background when an input is newly selected.
- 0.10 m - Yellow background when you change a parameter from the default value.
- 1 - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

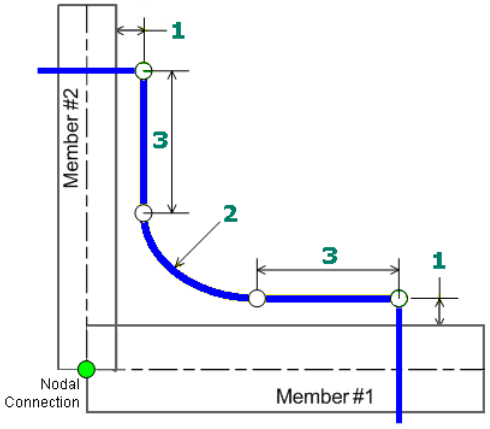
Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

If the angle between the members is less than 175 degrees, the following free edge is created:



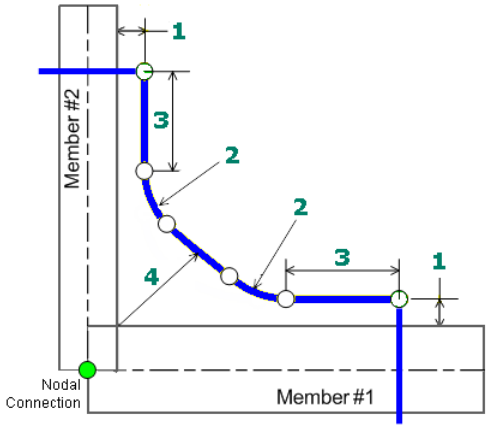
1 - Weld toe

2 - Fillet

3 - Cutback

4 - Arm length

If the angle between the members is between 175 and 180 degrees, then the following free edge is created:



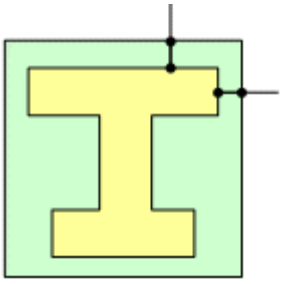
Weld Toe - Specifies the offset distance between the edges of the members and the side plate free edges.

Fillet - Specifies the radius of the fillet between two successive line segments of the side plate free edge.

Arm Length - Specifies the distance between the line perpendicular to the bisector of the two member axes and the intersection point between the two member edges. This parameter is only used if the angle between the members is between 175 and 180 degrees.

Cutback - Specifies the distance to cut back the incoming members away from the fillet of the side plate free edge.

Extension - Specifies the distance to extend the trimming planes. If the macro does not properly extend the trimming planes through all of the member cross-section, you can increase this value.



Side - Specifies the flange location of the side plate. Select **Front** or **Back**.

What do you want to do?

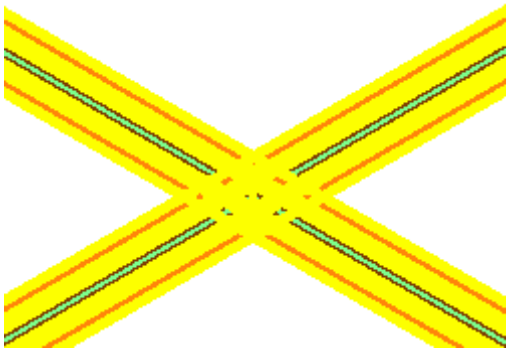
- *Place a side plate* (on page 288)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a side plate

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

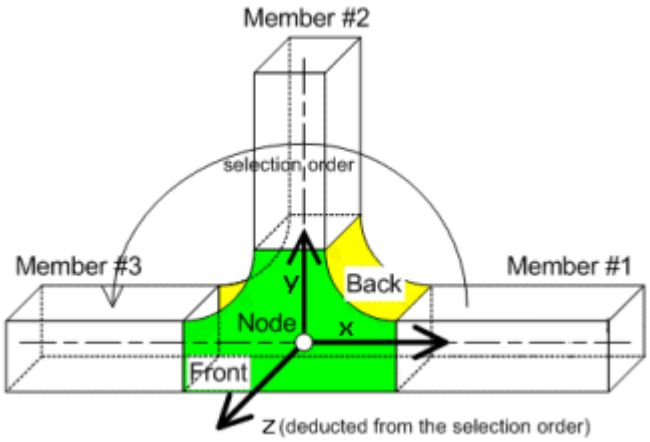
   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Side Plates** tab, click **SidePlate**.

   *The **Side Plate** ribbon displays.*
4. In a graphic view, select the co-planar members that support the side plate. Select the members in clockwise or counter-clockwise consecutive order.



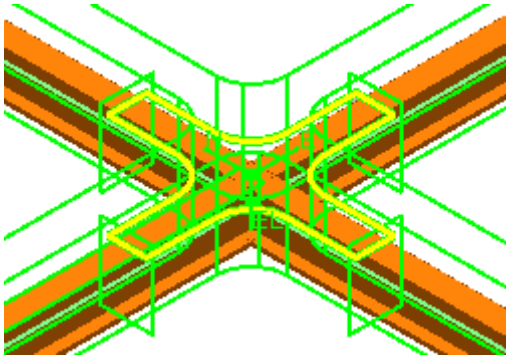
TIPS

- The first member that you select defines the X-axis for the side plate.
- The second member that you select defines the Y-axis for the side plate.



5. Type the required values for **Weld Toe**, **Fillet**, **Arm Length**, and **Cutback**.
6. Select the flange location of the side plate from the **Side** list.
7. Click **Continue**.

   *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



8. If the plate preview does not display on the correct member flange side, then:
   a. Click **Select Supports** .

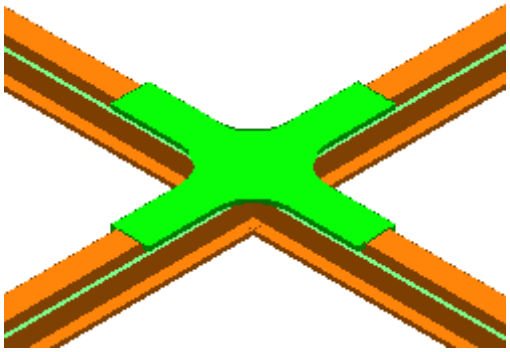
      *The ribbon display returns to the **Side Plate** ribbon.*

   b. Select the opposite location from the **Side** list, click **Accept** , and then click **Continue**.

      *A preview of the plate, now on the opposite member flanges, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

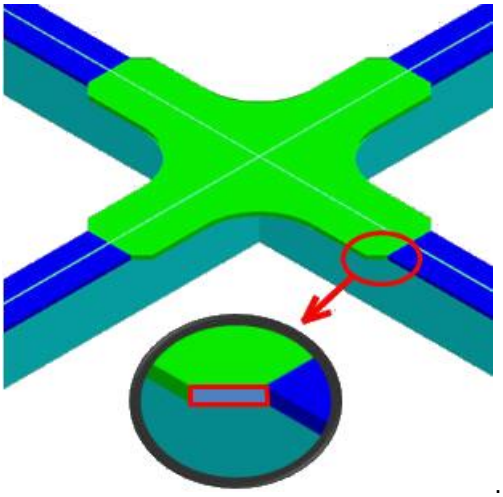
9. If needed, click **Define Boundaries** , and define boundaries for the plate.
10. Click **Finish**.

The software creates the side plate.



TIP If the macro does not properly cut through the member cross-sections, type a new distance to extend the trimming planes in the **Extension** box.

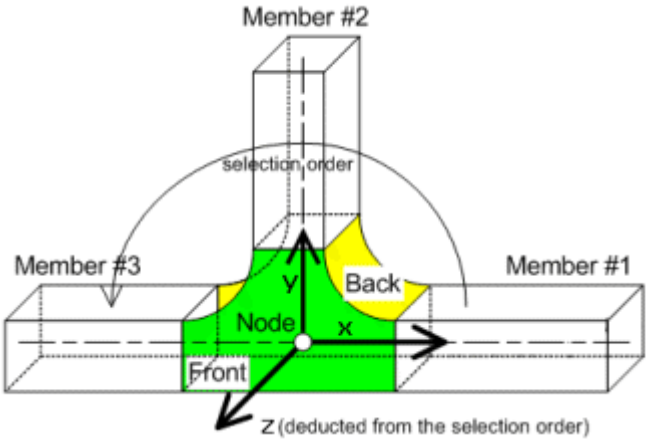
Side Plate 2



Tooltip: SidePlate2

Creates a side plate by connecting all flanges on a set of incoming members that share a nodal connection. The corners of the side plate are sniped.

The physical axes of the members are end-point connected and coplanar. The common end point is the node. You must have at least two members with non-collinear axes. The incoming flange and web plate parts are trimmed. For built-up members, the software trims the incoming flange and web parts by the edges of the side plate.



The macro computes a local coordinate system from the following:

- X-axis - Defined by the axis of the first selected member oriented from the nodal point to the other end of the member.
- Y-axis - Defined by the axis of the second selected member oriented from the nodal point to the other end of the member.
- Z-axis - Calculated as the cross-product of X-axis by Y-axis.

The type of free edge on the side plate depends on the angle between the axes of two adjacent members.

Side Plate 2 Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the primary members that support the side plate. These members can be rolled members or built-up members. Their physical axes must be coplanar and connected at the end points. Select the first two members in a clockwise or counter-clockwise direction. The first member that you select defines the X-axis for the side plate. The second member that you select defines the Y-axis for the side plate.

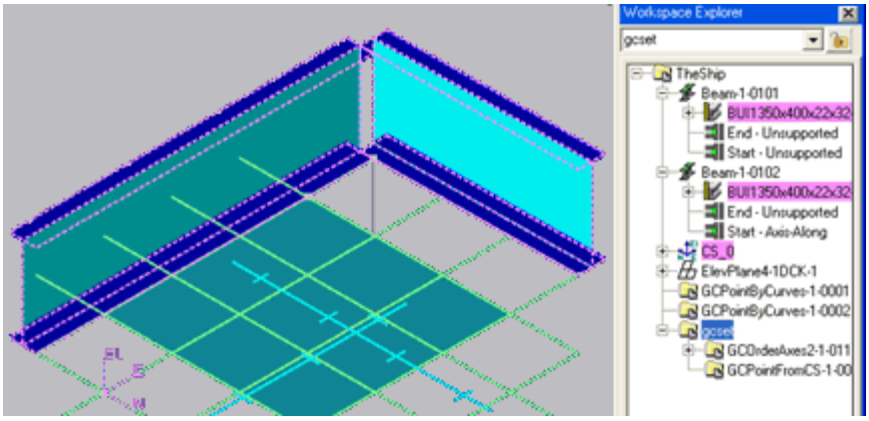
(2) - (Optional) Specifies secondary members that are attached to the face of the side plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

- 1 - No background color when no input is selected.
- 1 - Yellow background when an input is newly selected.
- 0.10 m - Yellow background when you change a parameter from the default value.
- 1 - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

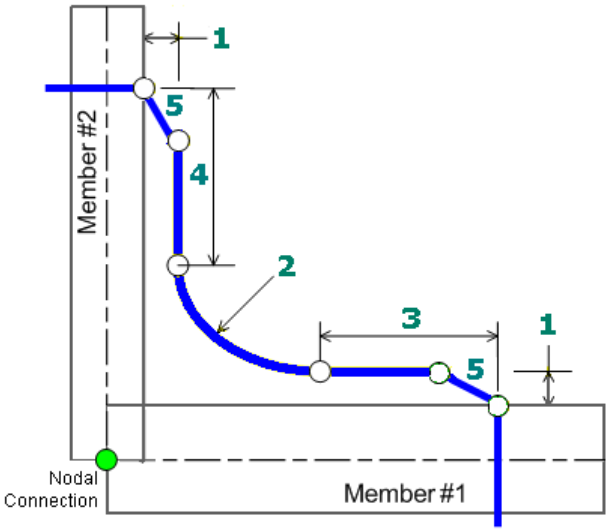
Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.



1 - Weld toe

2 - Fillet

3 - Cutback

4 - Cutback 2

5 - Slope ratio

Weld Toe - Specifies the offset distance between the edges of the members and the side plate free edges.

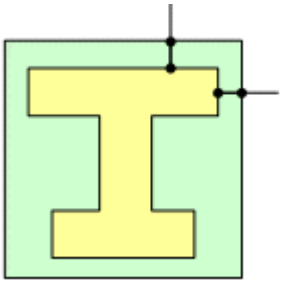
Fillet - Specifies the radius of the fillet between two successive line segments of the side plate free edge.

Cutback - Specifies the distance to cut back the incoming members away from the fillet of the side plate free edge.

Cutback 2 - Specifies the distance to cut back the second line of incoming members away from the fillet of the side plate free edge.

Slope Ratio - Specifies the slope of sniped corners at the side plate free edges.

Extension - Specifies the distance to extend the trimming planes. If the macro does not properly extend the trimming planes through all of the member cross-section, you can increase this value.



Side - Specifies the flange location of the side plate. Select **Front** or **Back**.

What do you want to do?

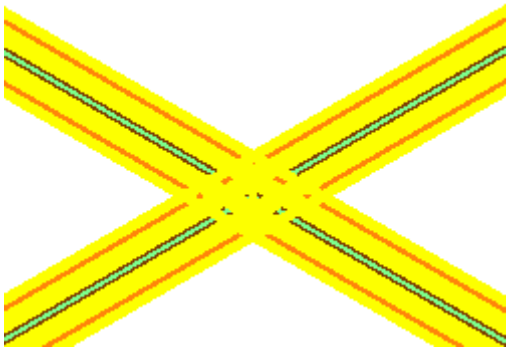
- *Place a side plate with sniped corners* (on page 294)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a side plate with sniped corners

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

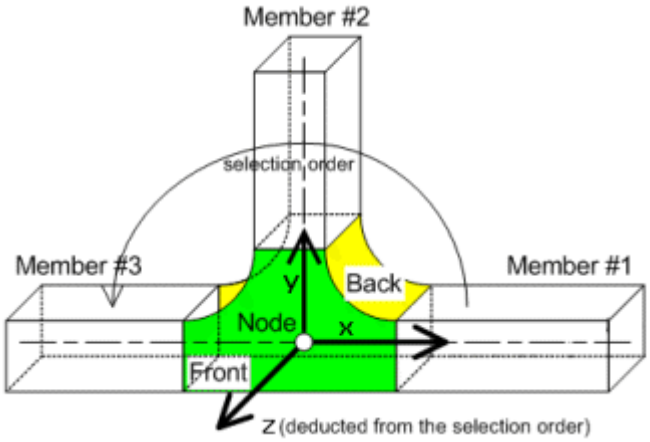
   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Side Plates** tab, click **SidePlate2**.

   *The **Side Plate 2** ribbon displays.*
4. In a graphic view, select the co-planar members that support the side plate. Select the members in clockwise or counter-clockwise consecutive order.



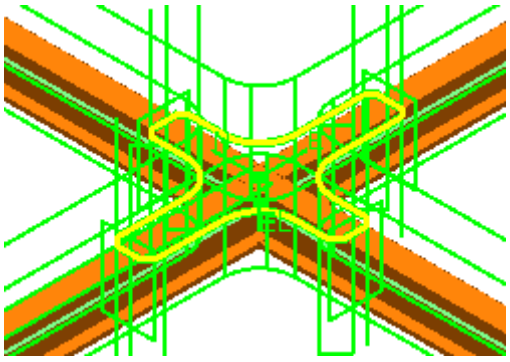
TIPS

- The first member that you select defines the X-axis for the side plate.
- The second member that you select defines the Y-axis for the side plate.



5. Type the required values for **Weld Toe**, **Fillet**, **Slope Ratio**, **Cutback**, and **Cutback 2**.
6. Select the flange location of the side plate from the **Side** list.
7. Click **Continue**.

*A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



8. If the plate preview does not display on the correct member flange side, then:

   a. Click **Select Supports** .

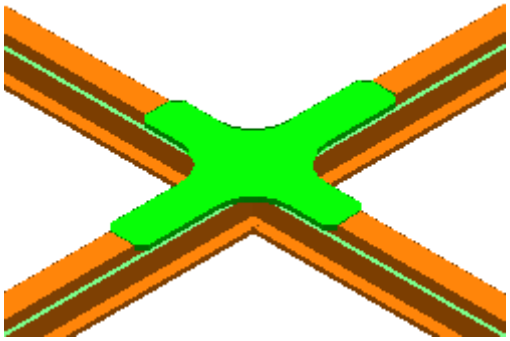
      *The ribbon display returns to the **Side Plate 2** ribbon.*

   b. Select the opposite location from the **Side** list, click **Accept** , and then click **Continue**.

      *A preview of the plate, now on the opposite member flanges, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

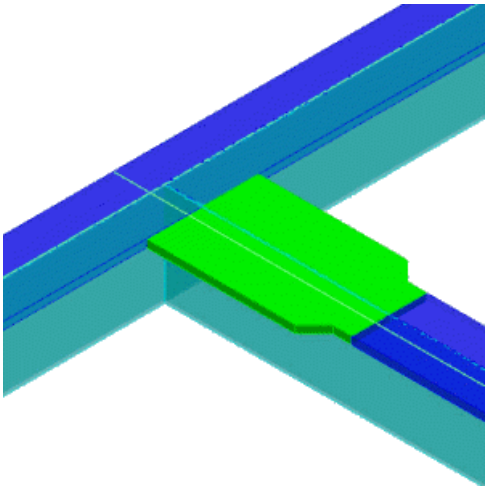
9. If needed, click **Define Boundaries** , and define boundaries for the plate.
10. Click **Finish**.

    The software creates the side plate.



TIP If the macro does not properly cut through the member cross-sections, type a new distance to extend the trimming planes in the **Extension** box.

Side Plate Axial



Tooltip: SidePlateAxial

Creates a side plate by connecting the flange on an incoming intercostal supported member that share a nodal connection with a continuous supporting member.

The physical axes of the members are coplanar. The end point of the supported member is the node with the supporting member. The software trims the incoming flange and web plate parts of the supported member. For built-up members, the software trims the incoming flange and web parts of the supported member by the edges of the side plate.

Side Plate Axial Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the supporting member. This member can be a rolled member or a built-up member.

2 - Specifies the supported member. This member can be a rolled member or a built-up member. The physical axis must be coplanar with and nodally connected to the supporting member.

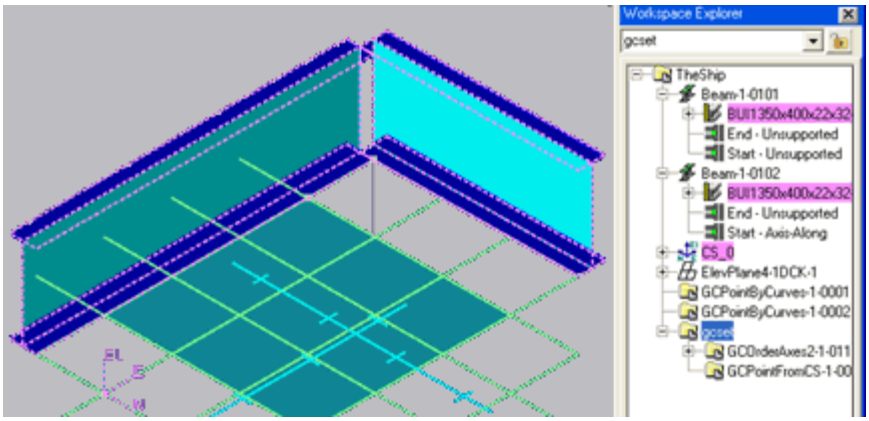
(3) - (Optional) Specifies secondary members that are attached to the face of the side plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

- 1 - No background color when no input is selected.
- 1 - Yellow background when an input is newly selected.
- 0.10 m - Yellow background when you change a parameter from the default value.
- 1 - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

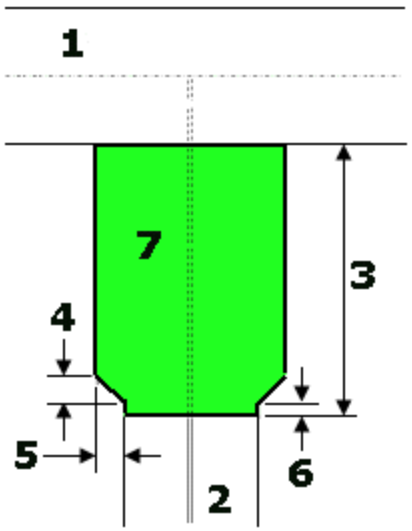
Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.



1 - Supporting member

2 - Supported member

3 - Length

4 - Chamfer length

5 - Chamfer width

6 - Setback

7 - Axial side plate

Length - Specifies the length of the side plate.

Chamfer Length - Specifies the length of sniped corners at the free edges of the side plate.

Chamfer Width - Specifies the width of sniped corners at the free edges of the side plate.

Setback - Specifies the distance to set back sniped corners from the end of the side plate.

Location - Specifies the flange on which to place the insert plate relative to the member major axis. Select **Top** to place the insert plate on the top flanges of the members. Select **Bottom** to place the insert plate on the bottom flanges of the members.

What do you want to do?

- *Place an axial side plate* (on page 298)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

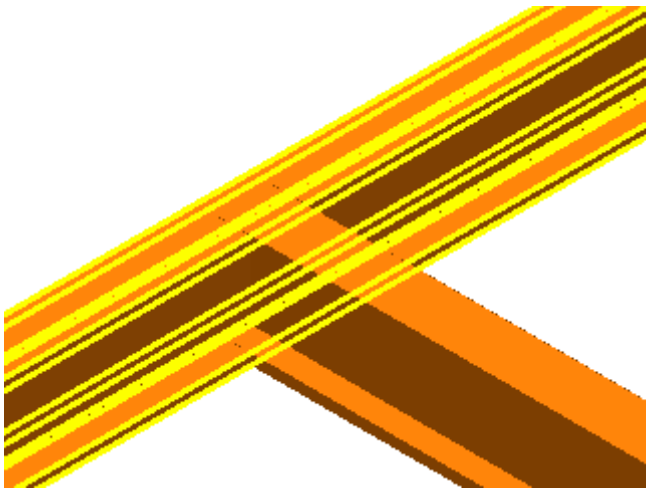
Place an axial side plate

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

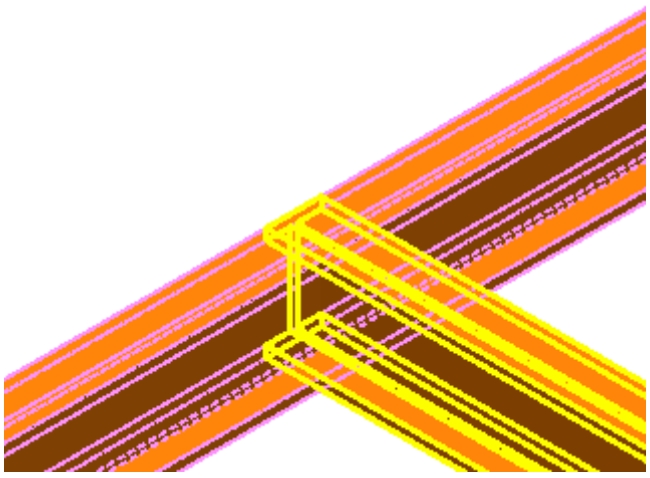
   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Side Plates** tab, click **SidePlateAxial**.

   *The **Side Plate Axial** ribbon displays.*

4. In a graphic view, select the supporting member.

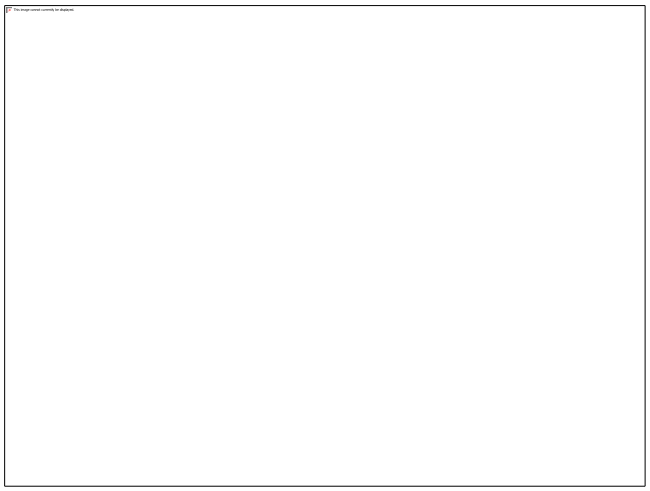


5. Select the supported member.



6. Type values for **Length**, **Chamfer Length**, **Chamfer Width**, and **Setback**.
7. Select the location of the side plate from the **Side** list.
8. Click **Continue**.

*A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



9. If the plate preview does not display on the correct member flange side, then:
   a. Click **Select Supports** .

      *The ribbon display returns to the **Side Plate Axial** ribbon.*

   b. Select the opposite location from the **Side** list, click **Accept** , and then click **Continue**.

      *A preview of the plate, now on the opposite member flange, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

10. If needed, click **Define Boundaries** , and define boundaries for the plate.
11. Click **Finish**.

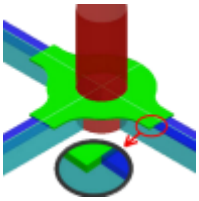
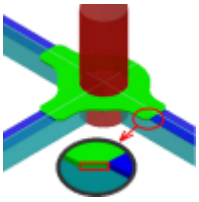
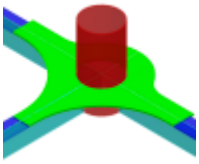
    The software creates the side plate.



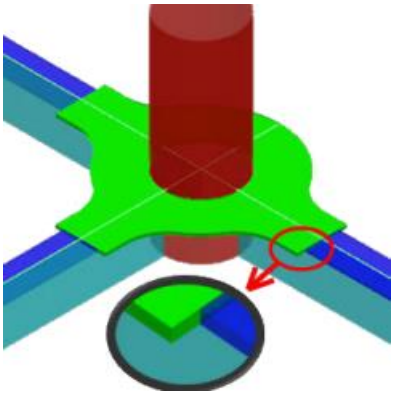
TIP If the macro does not properly cut through the member cross-sections, type a new distance to extend the trimming planes in the **Extension** box.

Ring Plates Tab (Advanced Plate Palette Dialog Box)

The following advanced plate system macros create ring plates, which reinforce one or more members sharing a nodal connection and penetrated by a perpendicular cylindrical member. A nodal connection is typically a construction assembly between built-up members and consists of side plates or ring plates, transition plates, and continuity plates.

	**Ring Plate 1** - Creates a ring plate by connecting all flanges on a set of incoming members that share a nodal connection and a penetrating perpendicular cylindrical member. The corners of the ring plate are square. For more information, see *Ring Plate 1* (on page 301).
	**Ring Plate 2** - Creates a ring plate by connecting all flanges on a set of incoming members that share a nodal connection and a penetrating perpendicular cylindrical member. The corners of the ring plate are sniped. For more information, see *Ring Plate 2* (on page 308).
	**Ring Plate 3**- Creates a ring plate by connecting all flanges on a set of incoming members that share a nodal connection and a penetrating perpendicular cylindrical member. The ring plate has a continuous radial edge between members. For more information, see *Ring Plate 3* (on page 315).

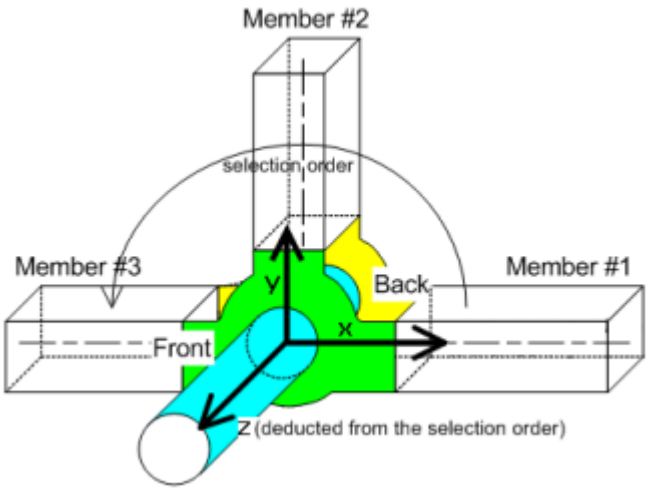
Ring Plate 1



Tooltip: RingPlate

Creates a ring plate by connecting all flanges on a set of incoming members that share a nodal connection and a penetrating perpendicular cylindrical member. The corners of the ring plate are square.

The physical axes of the incoming members are end-point connected and coplanar. The common end point is the node. The axis of the perpendicular member traverses the node. A ring plate can be defined by a single incoming member and the perpendicular member. The incoming flange and web plate parts are trimmed. For built-up members, the incoming flange and web parts are trimmed by the edges of the ring plate.



The macro computes a local coordinate system from the following:

1. If the ring plate has one incoming member:
   - X-axis - Defined by the axis of the member oriented from the nodal point to the other end of the member.
   - Z-axis - Defined by the axis of the perpendicular cylindrical member oriented from the nodal point to the other end of the perpendicular member.
   - Y-axis - Calculated as the cross product of Z-axis by Y-axis.
2. If the ring plate has more than one incoming member:
   - X-axis - Defined by the axis of the first located member oriented from the nodal point to the other end of the member.
   - Y-axis - Defined by the axis of the second located member oriented from the nodal point to the other end of the member.
   - Z-axis - Calculated as the cross product of X-axis by Y-axis.

Ring Plate 1 Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1

Specifies the members that support the ring plate. These members can be rolled or built-up members. Their physical axes must be coplanar and connected at the end points. Select the first two members in a clockwise or counter-clockwise direction. The first member that you select defines the X-axis for the ring plate. If you select a second member, that member defines the Y-axis for the ring plate. If there is only a single incoming member, then the macro computes the Y-axis based on the supporting member and the perpendicular member.

2

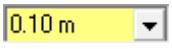
Specifies the tubular member that penetrates the ring plate. This member can be a rolled or built-up member, but it must be orthogonal to the plane described by the members selected in step **1**, and it must pass through the node.

(3)

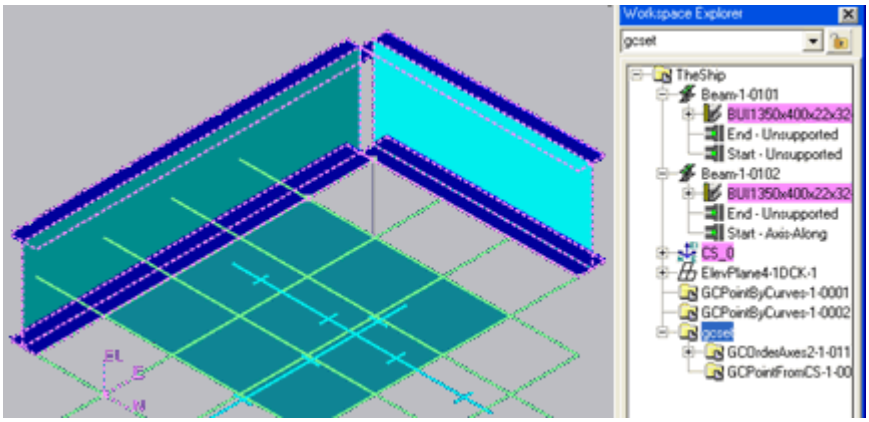
(Optional) Specifies secondary members that are attached to the face of the ring plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

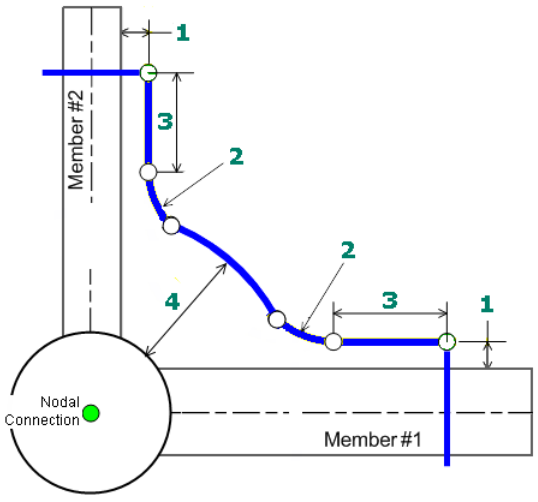
Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.



1 - Weld toe

2 - Fillet

3 - Cutback

4 - Offset

Weld Toe

Specifies the offset distance between the edges of the members and the ring plate free edges.

Offset

Specifies the offset between the extracted cylinder and the outer circle of the ring plate free edges.

Fillet

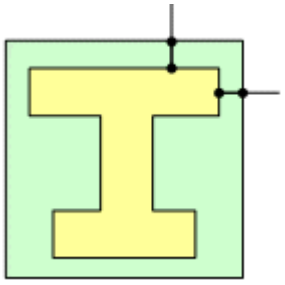
Specifies the radius of the fillet between the offset radius and the cutback length of the free edge ring plate.

Cutback

Specifies the distance to cut back the incoming members from the fillet.

Extension

Specifies the distance to extend the trimming planes. If the macro does not properly extend the trimming planes through all of the member cross-section, you can increase this value.



Support

Specifies the flange location of the ring plate. Select **Front** or **Back**.

Is Cut By Tube

Specifies whether a connection is created between the tubular member or the ring plate. Select **True** for a connection. Select **False** for no connection.

What do you want to do?

- *Place a ring plate* (on page 305)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

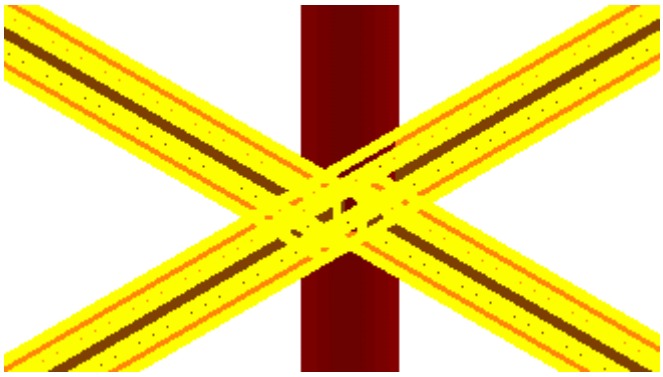
Place a ring plate

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Side Plates** tab, click **RingPlate1**.

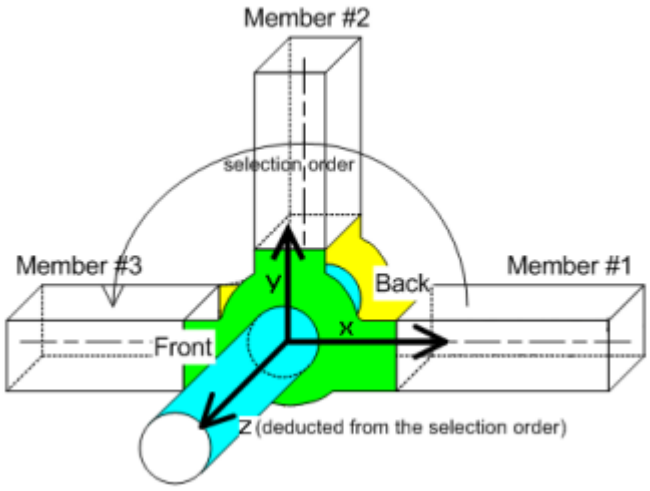
   *The **Ring Plate 1** ribbon displays.*

4. In a graphic view, select the co-planar members, in clockwise or counter-clockwise consecutive order, that support the ring plate.

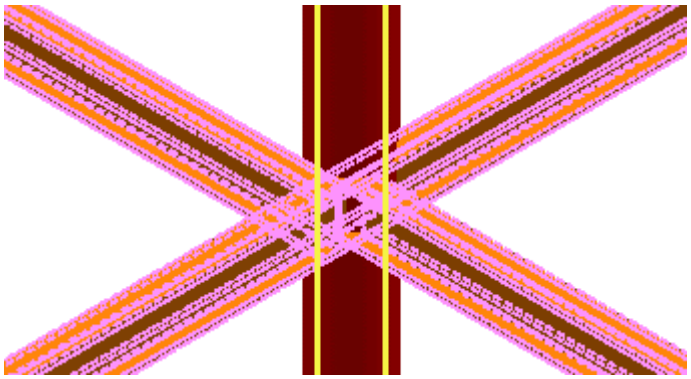


TIPS

- The first member that you select defines the X-axis for the ring plate.
- The second member that you select defines the Y-axis for the ring plate.

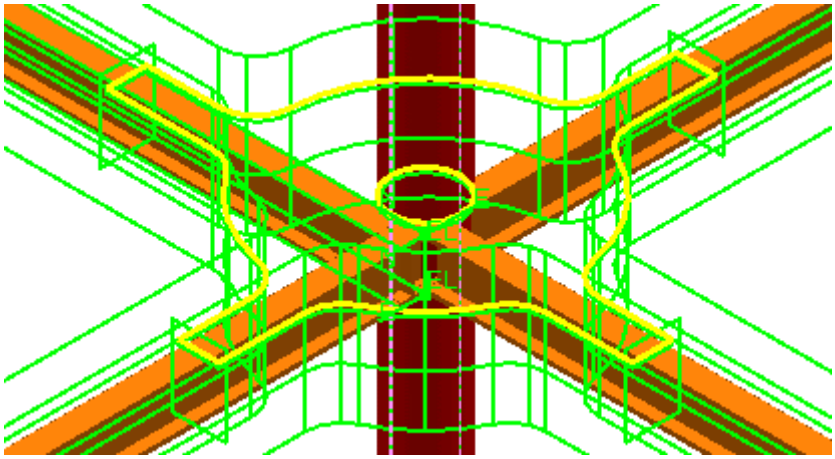


5. Click **2** on the ribbon.
6. Select the tubular member that penetrates the ring plate. This can be a rolled or built-up member. The member must be orthogonal to the plane defined by the members selected for **1**, and it must pass through the node point.



7. Type the required values for **Weld Toe**, **Offset**, **Fillet**, and **Cutback**.
8. Select the flange location of the ring plate in the **Support** list.
9. Select the connection of the ring plate in the **Is Cut By Tube** list.
10. Click **Continue**.

    *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



11. If the plate preview does not display on the correct member flange side, then:
    a. Click **Select Supports** .

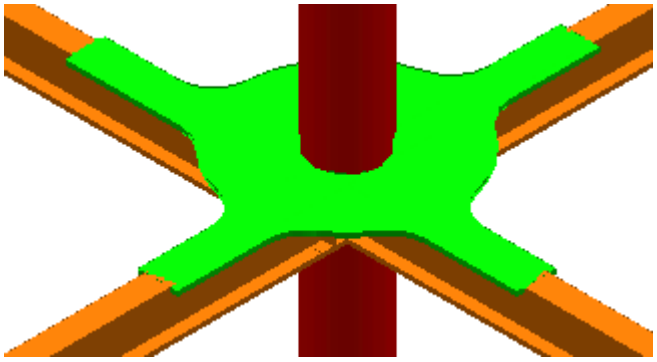
       *The ribbon display returns to the **Ring Plate 1** ribbon.*

    b. Select the opposite location from the **Support** list, click **Accept** , and then click **Continue**.

       *A preview of the plate, now on the opposite member flanges, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

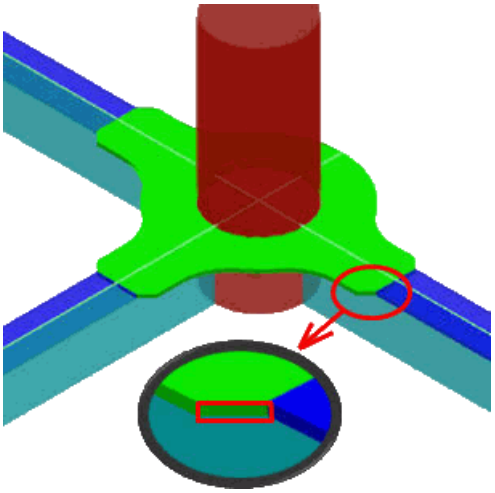
12. If needed, click **Define Boundaries** , and define boundaries for the plate.
13. Click **Finish**.

    The software creates the ring plate.



TIP If the macro does not properly cut through the member cross-sections, type a new distance to extend the trimming planes in the **Extension** box.

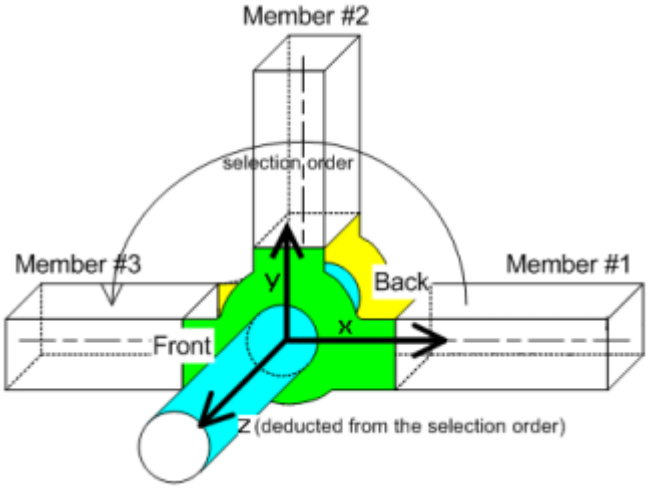
Ring Plate 2



Tooltip: RingPlate2

Creates a ring plate by connecting all flanges on a set of incoming members that share a nodal connection and a penetrating perpendicular cylindrical member. The corners of the ring plate are sniped.

The physical axes of the incoming members are end-point connected and coplanar. The common end point is the node. The axis of the perpendicular member traverses the node. A ring plate can be defined by a single incoming member and the perpendicular member. The incoming flange and web plate parts are trimmed. For built-up members, the incoming flange and web parts are trimmed by the edges of the ring plate.



The macro computes a local coordinate system from the following:

1. If the ring plate has one incoming member:
   - X-axis - Defined by the axis of the member oriented from the nodal point to the other end of the member.
   - Z-axis - Defined by the axis of the perpendicular cylindrical member oriented from the nodal point to the other end of the perpendicular member.

- Y-axis - Calculated as the cross product of Z-axis by Y-axis.

2. If the ring plate has more than one incoming member
   - X-axis - Defined by the axis of the first located member oriented from the nodal point to the other end of the member.
   - Y-axis - Defined by the axis of the second located member oriented from the nodal point to the other end of the member.
   - Z-axis - Calculated as the cross product of X-axis by Y-axis.

Ring Plate 2 Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1

Specifies the members that support the ring plate. These members can be rolled or built-up members. Their physical axes must be coplanar and connected at the end points. Select the first two members in a clockwise or counter-clockwise direction. The first member that you select defines the X-axis for the ring plate. If you select a second member, that member defines the Y-axis for the ring plate. If there is only a single incoming member, then the macro computes the Y-axis based on the supporting member and the perpendicular member.

2

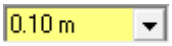
Specifies the tubular member that penetrates the ring plate. This member can be a rolled or built-up member, but it must be orthogonal to the plane described by the members selected in step **1**, and it must pass through the node.

(3)

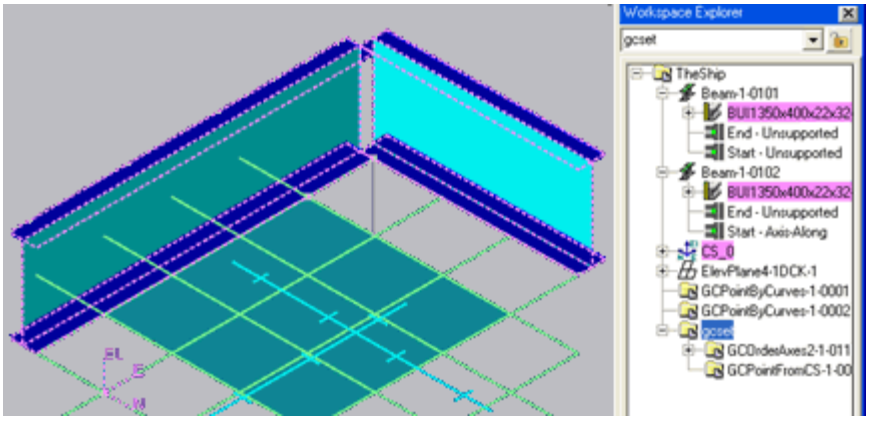
(Optional) Specifies secondary members that are attached to the face of the side plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

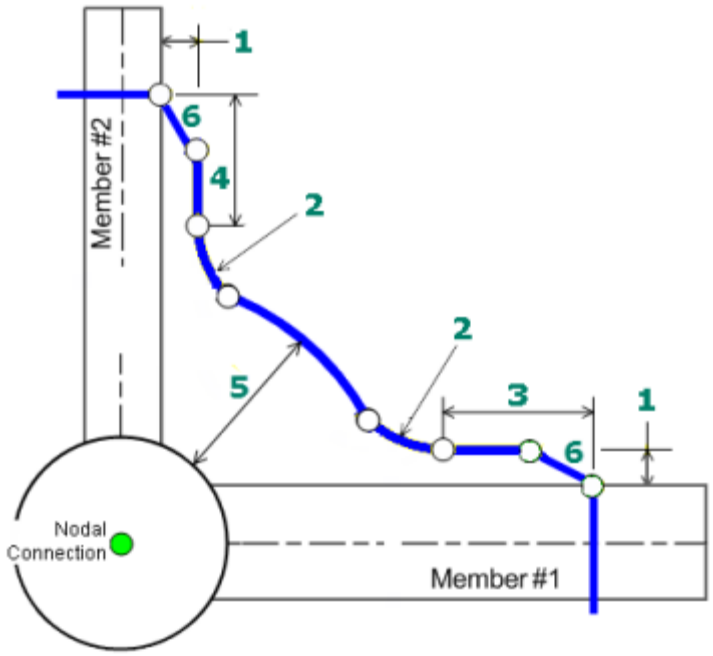
Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.



1 - Weld toe

2 - Fillet

3 - Cutback

4 - Cutback 2

5 - Offset

6 - Slope Ratio

Weld Toe

Specifies the offset distance between the edges of the members and the ring plate free edges.

Offset

Specifies the offset between the extracted cylinder and the outer circle of the ring plate free edges.

Fillet

Specifies the radius of the fillet between the offset radius and the cutback length of the ring plate free edge.

Cutback

Specifies the distance to cut back the incoming members from the fillet.

Cutback 2

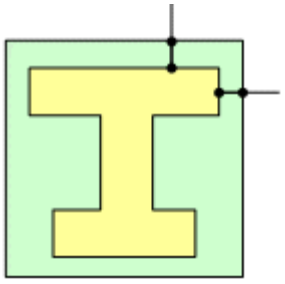
Specifies the distance to cut back the second line of incoming members from the fillet.

Slope Ratio

Specifies the slope of sniped corners at the ring plate free edges.

Extension

Specifies the distance to extend the trimming planes. If the macro does not properly extend the trimming planes through all of the member cross-section, you can increase this value.



Support

Specifies the location of the ring plate. Select **Front** or **Back**.

Is Cut By Tube

Specifies whether a connection is created between the tubular member or the ring plate. Select **True** for a connection. Select **False** for no connection.

What do you want to do?

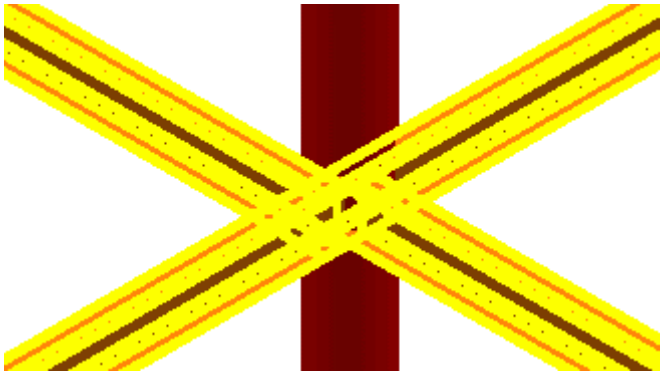
- *Place a ring plate with sniped corners* (on page 312)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a ring plate with sniped corners

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

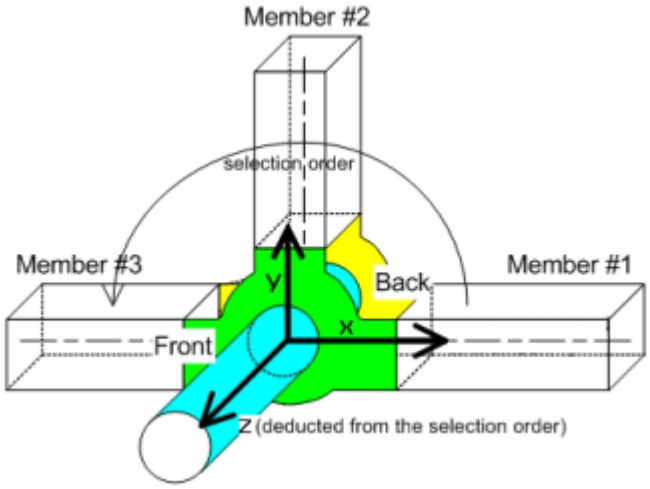
   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Side Plates** tab, click **RingPlate2**.

   *The **Ring Plate 2** ribbon displays.*
4. In a graphic view, select the co-planar members, n clockwise or counter-clockwise consecutive order, that support the ring plate.

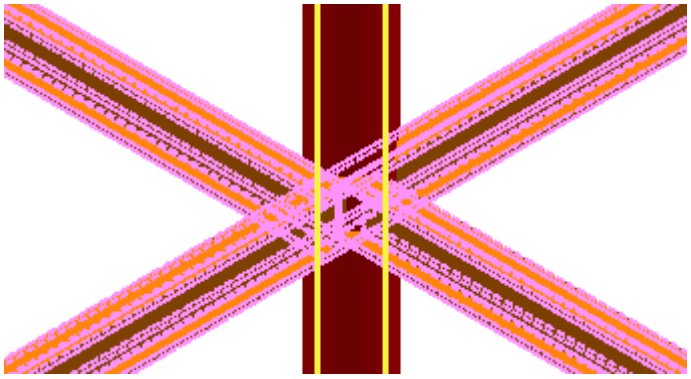


TIPS

- The first member that you select defines the X-axis for the ring plate.
- The second member that you select defines the Y-axis for the ring plate.

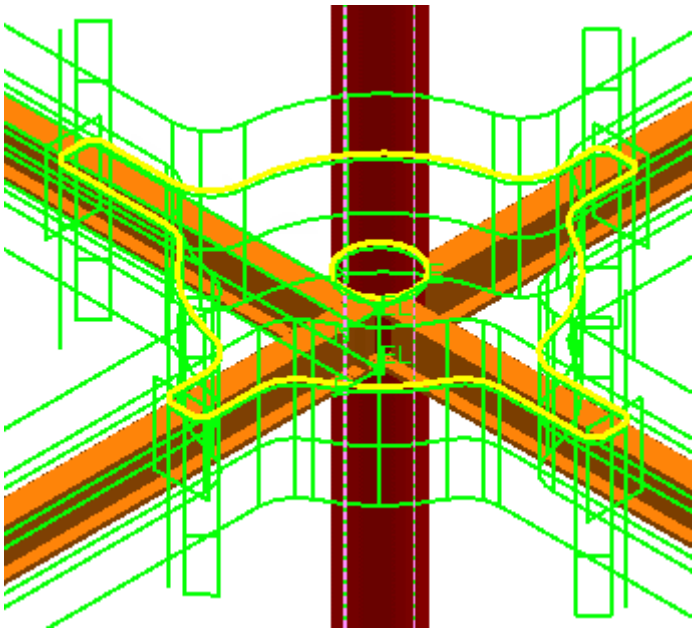


5. Click **2** on the ribbon.
6. Select the tubular member that penetrates the ring plate. This member can be a rolled or built-up member. It must be orthogonal to the plane defined by the members selected for **1**, and it must pass through the node point.



7. Type the required values for **Weld Toe**, **Offset**, **Fillet**, **Slope Ratio**, **Cutback**, and **Cutback 2**.
8. Select the flange location of the ring plate in the **Support** list.
9. Select the continuity of the ring plate in the **Is Cut By Tube** list.
10. Click **Continue**.

*A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



11. If the plate preview does not display on the correct member flange side, then:
    a. Click **Select Supports** .

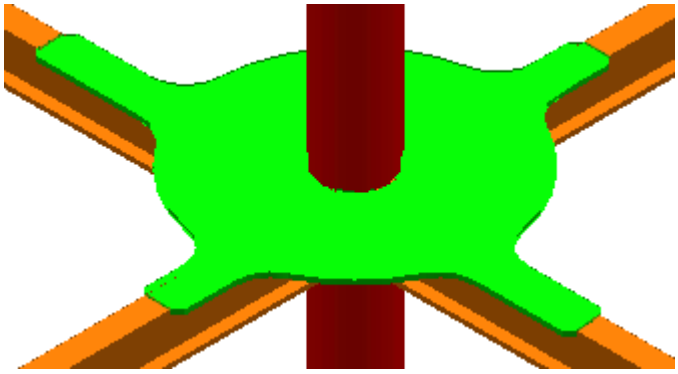
       *The ribbon display returns to the **Ring Plate 2** ribbon.*

    b. Select the opposite location from the **Support** list, click **Accept** , and then click **Continue**.

       *A preview of the plate, now on the opposite member flanges, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

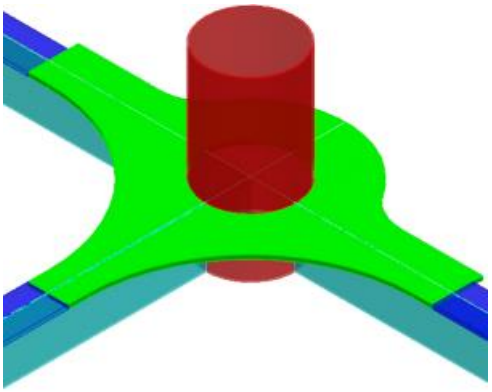
12. If needed, click **Define Boundaries** , and define boundaries for the plate.
13. Click **Finish**.

The software creates the ring plate.



TIP If the macro does not properly cut through the member cross-sections, type a new distance to extend the trimming planes in the **Extension** box.

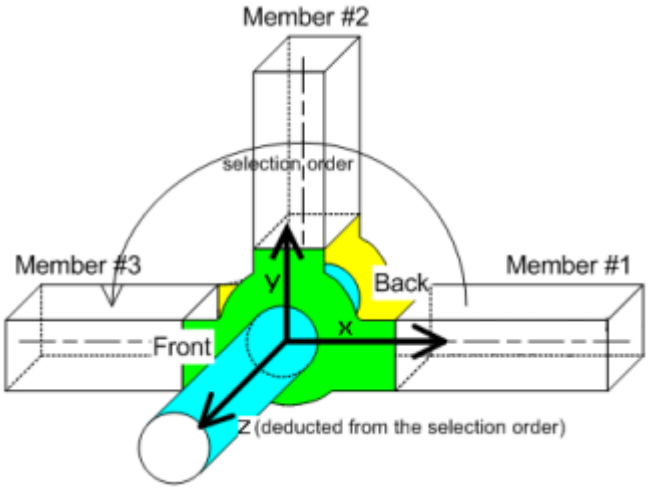
Ring Plate 3



Tooltip: RingPlate3

Creates a ring plate by connecting all flanges on a set of incoming members that share a nodal connection and a penetrating perpendicular cylindrical member. The ring plate has a large fillet radius between non-colinear incoming members.

The physical axes of the incoming members are end-point connected and coplanar. The common end point is the node. The axis of the perpendicular member traverses the node. A ring plate can be defined by a single incoming member and the perpendicular member. The incoming flange and web plate parts are trimmed. For built-up members, the incoming flange and web parts are trimmed by the edges of the ring plate.



The macro computes a local coordinate system from the following:

1. If the ring plate has one incoming member:
   - X-axis - Defined by the axis of the member oriented from the nodal point to the other end of the member.
   - Z-axis - Defined by the axis of the perpendicular cylindrical member oriented from the nodal point to the other end of the perpendicular member.
   - Y-axis - Calculated as the cross product of Z-axis by Y-axis.
2. If the ring plate has more than one incoming member

- X-axis - Defined by the axis of the first located member oriented from the nodal point to the other end of the member.
- Y-axis - Defined by the axis of the second located member oriented from the nodal point to the other end of the member.
- Z-axis - Calculated as the cross product of X-axis by Y-axis.

Ring Plate 3 Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1

Specifies the members that support the ring plate. These members can be rolled or built-up members. Their physical axes must be coplanar and connected at the end points. Select the first two members in a clockwise or counter-clockwise direction. The first member that you select defines the X-axis for the ring plate. If you select a second member, that member defines the Y-axis for the ring plate. If there is only a single incoming member, then the macro computes the Y-axis based on the supporting member and the perpendicular member.

2

Specifies the tubular member that penetrates the ring plate. This member can be a rolled or built-up member. It must be orthogonal to the plane described by the members selected in step **1**, and it must pass through the node.

(3)

(Optional) Specifies secondary members that are attached to the face of the side plate.

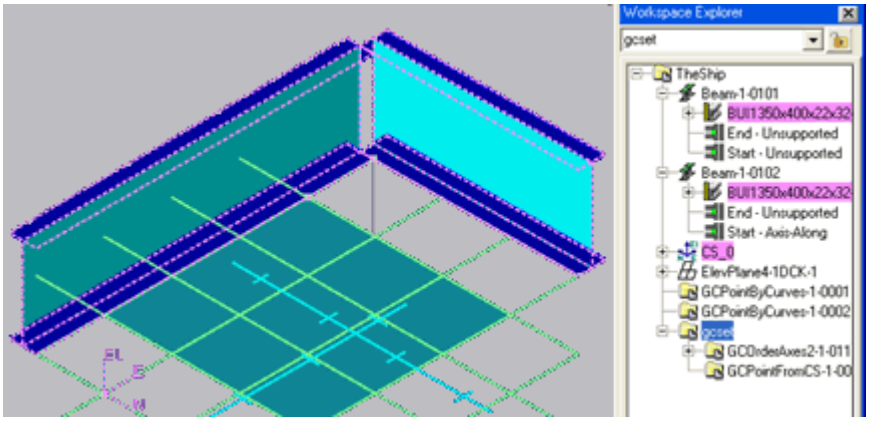
Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

- 1 - No background color when no input is selected.

-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

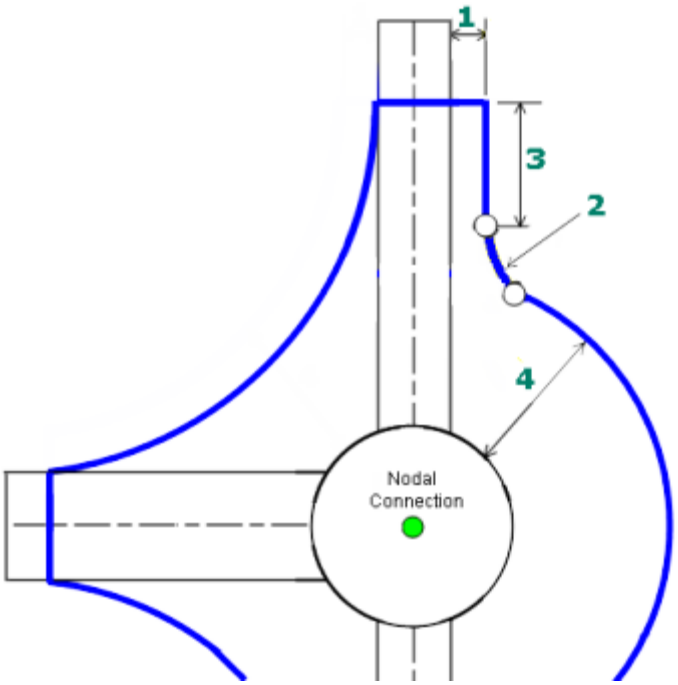
Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.



1 - Weld toe

2 - Fillet

3 - Cutback

4 - Offset

Weld Toe

Specifies the offset distance between the edges of the members and the ring plate free edges.

Offset

Specifies the offset between the extracted cylinder and the outer circle of the ring plate free edges.

Fillet

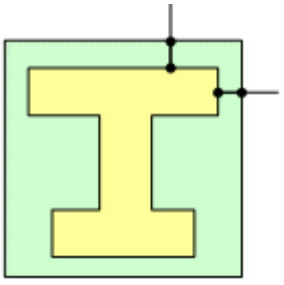
Specifies the radius of the fillet between the offset radius and the cutback length of the ring plate free edge.

Cutback

Specifies the distance to cut back the incoming members from the fillet.

Extension

Specifies the distance to extend the trimming planes. If the macro does not properly extend the trimming planes through all of the member cross-section, you can increase this value.



Support

Specifies the location of the ring plate. Select **Front** or **Back**.

Is Cut By Tube

Specifies whether a connection is created between the tubular member or the ring plate. Select **True** for a connection. Select **False** for no connection.

What do you want to do?

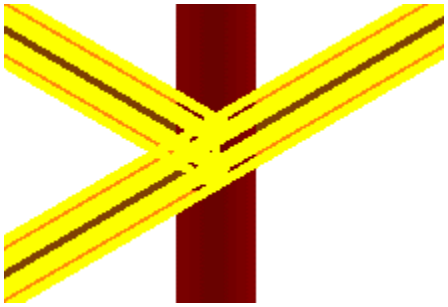
- *Place a ring plate with radial edges* (on page 319)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a ring plate with radial edges

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Side Plates** tab, click **RingPlate3**.

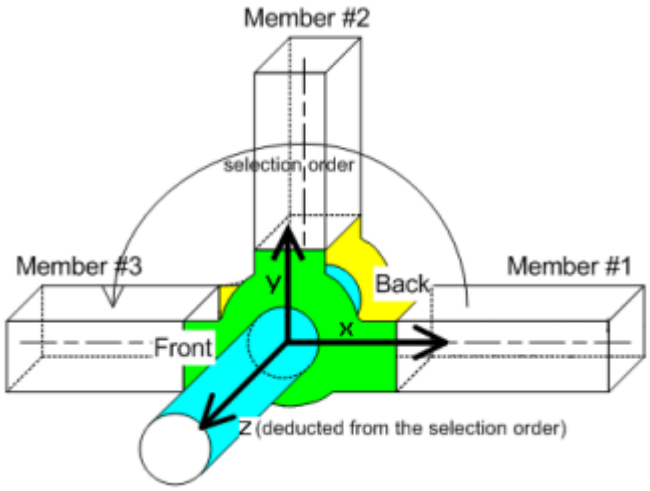
   *The **Ring Plate 3** ribbon displays.*
4. In a graphic view, select the co-planar members, in a clockwise or counter-clockwise consecutive order, that support the ring plate.



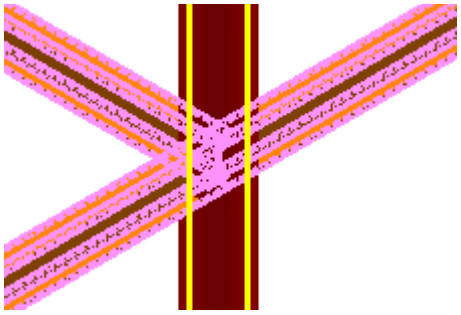
TIPS

- The first member that you select defines the X-axis for the ring plate.

- The second member that you select defines the Y-axis for the ring plate.

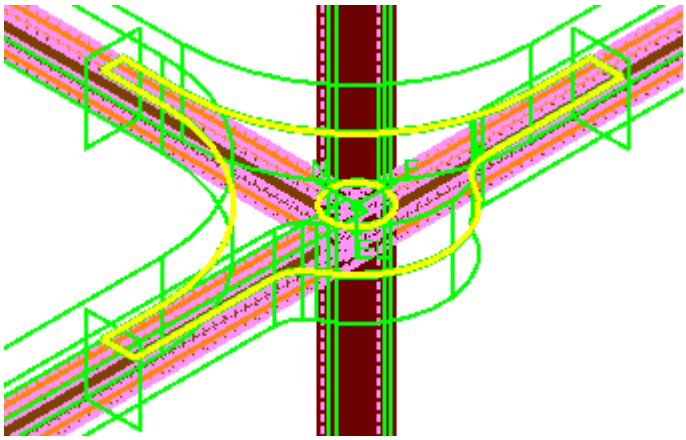


5. Click **2** on the ribbon.
6. Select the tubular member that penetrates the ring plate. This member can be a rolled or built up member. It must be orthogonal to the plane defined by the members selected for **1**, and it must pass through the node.



7. Type the required values for **Weld Toe**, **Offset**, **Fillet**, and **Cutback**.
8. Select the location of the ring plate in the **Support** list.
9. Select the continuity of the ring plate in the **Is Cut By Tube** list.
10. Click **Continue**.

*A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



11. If the plate preview does not display on the correct member flange side, then:
    a. Click **Select Supports** .

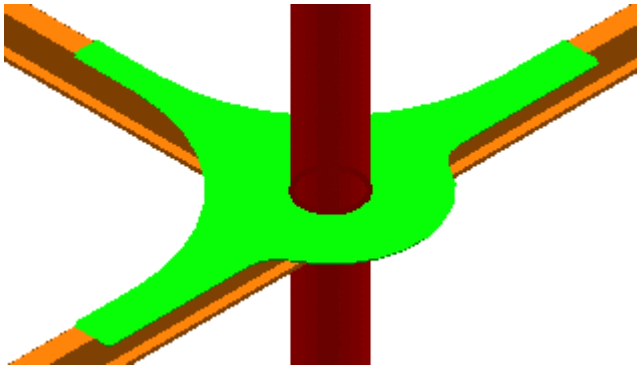
       *The ribbon display returns to the **Ring Plate 3** ribbon.*

    b. Select the opposite location from the **Support** list, click **Accept** , and then click **Continue**.

       *A preview of the plate, now on the opposite member flanges, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

12. If needed, click **Define Boundaries** , and define boundaries for the plate.
13. Click **Finish**.

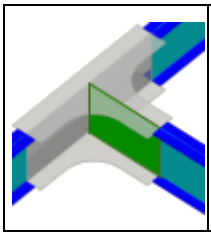
    The software creates the ring plate.



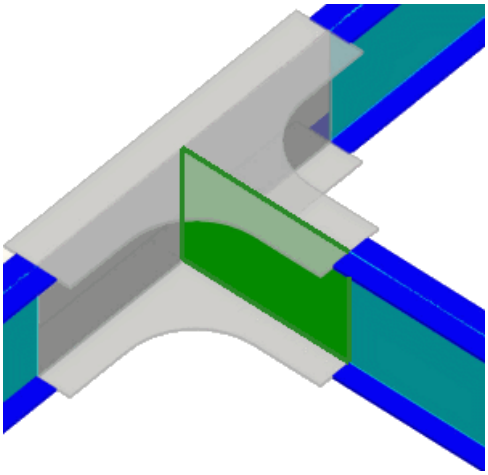
TIP If the macro does not properly cut through the member cross-sections, type a new distance to extend the trimming planes in the **Extension** box.

Continuity Plates Tab (Advanced Plate Palette Dialog Box)

The following advanced plate system macro creates continuity plates between side or ring plates and along the webs of supported members. Continuity plates are typically used to create offshore nodal connections between built-up members in combination with transition plates, sides plates, and ring plates.

	**Continuity Plate** - Creates a plate along supported member webs at the connection with supporting members and between top and bottom side plates or ring plates. For more information, see *Continuity Plate* (on page 322).

Continuity Plate



Tooltip: ContinuityPlate

Creates a plate along supported member webs at the connection with supporting members and between top and bottom side plates or ring plates. Continuity plates are typically used to create offshore nodal connections between built-up members in combination with transition plates, sides plates, and ring plates.

Continuity Plate Ribbon

Properties

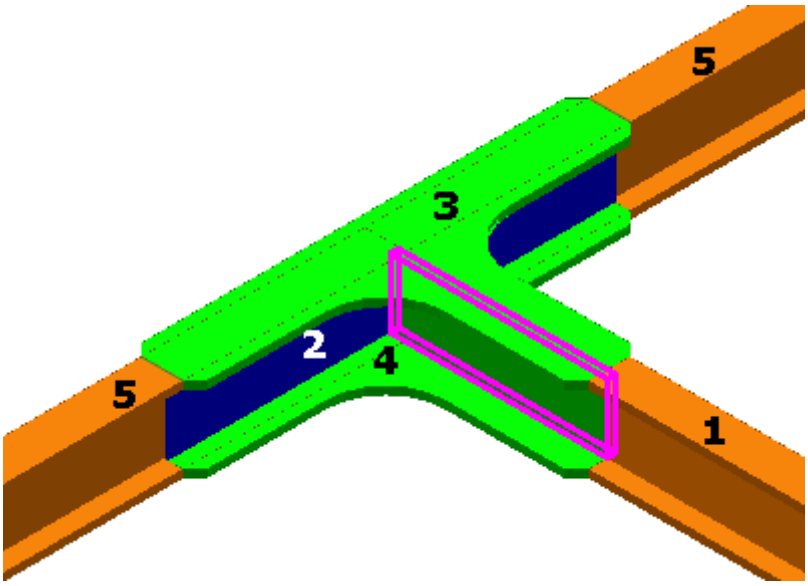
Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).

- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).



1 - Specifies the supported member with which to establish the continuity.

2 - Specifies the transition plate with which to limit the length of the continuity plate.

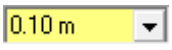
3 - Specifies the first bounding side or ring plate.

4 - Specifies the second bounding side or ring plate.

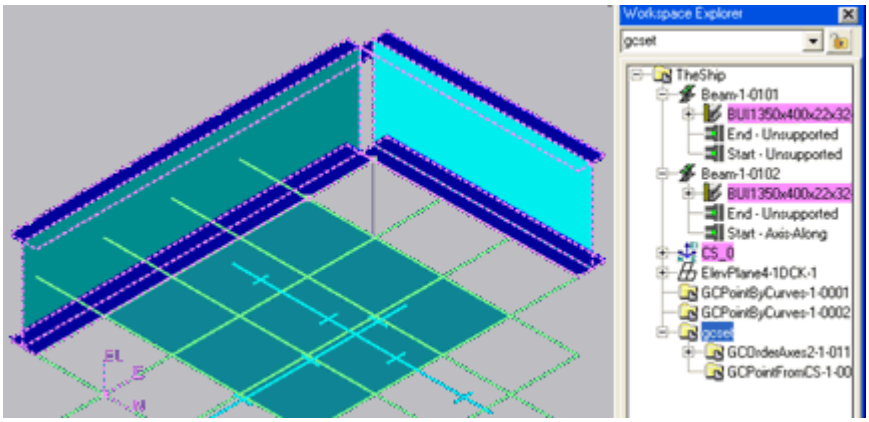
5 - Specifies the supporting member to use as a reference for the **Location** parameter. In the above figure, you can select either supporting member.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

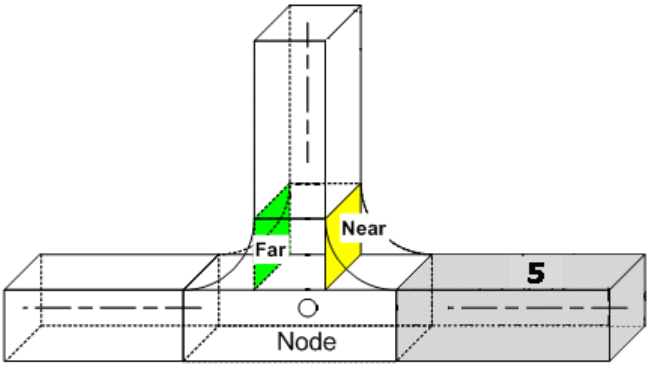
Accepts the selection for the current step.

Continue

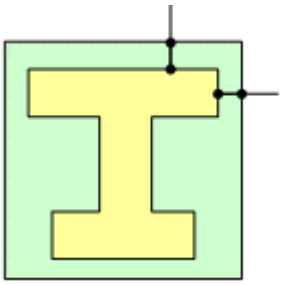
Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Location - Specifies the location of the continuity plate with respect to the selected reference member **5**. Select **Near**, **Far**, or **Centered**.



Extension - Specifies the distance to extend the trimming planes. If the macro does not properly extend the trimming planes through all of the member cross-section, you can increase this value.



What do you want to do?

- *Place a continuity plate* (on page 325)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a continuity plate

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. At the nodal connection, create two ring plates or side plates.

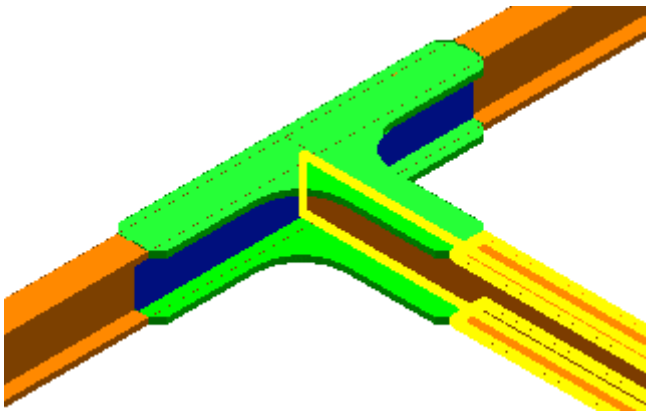
   Place a ring plate (on page 305)

   Place a side plate (on page 288)
4. At the nodal connection, create a transition plate.

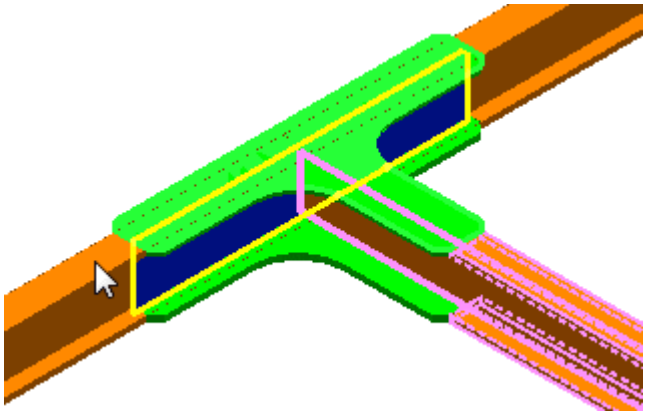
   Place a transition plate (on page 332)
5. On the **Continuity Plates** tab, click **ContinuityPlate**.

   *The **Continuity Plate** ribbon displays.*

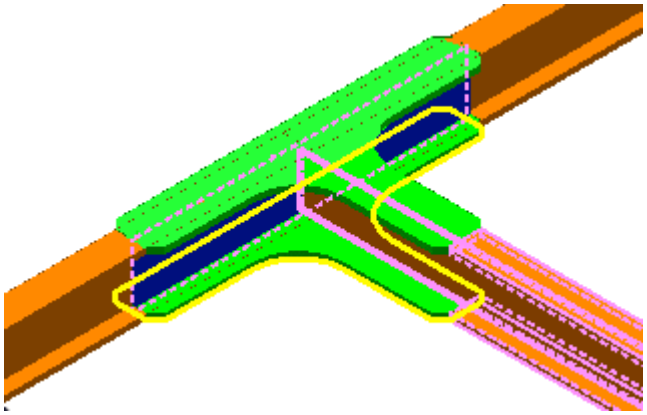
6. In a graphic view, select the supported member.



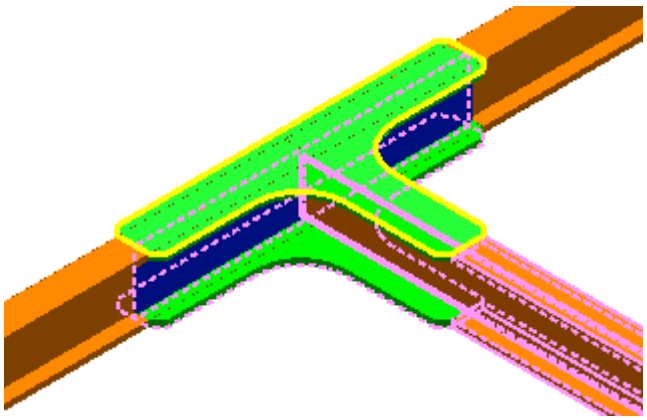
7. Select the transition plate.



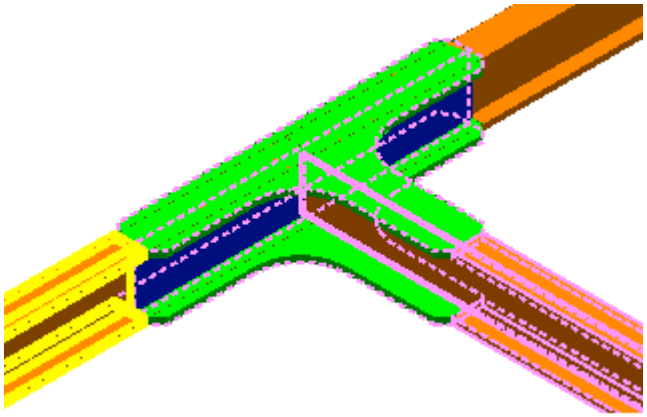
8. Select the first side or ring plate.



9. Select the second side or ring plate.

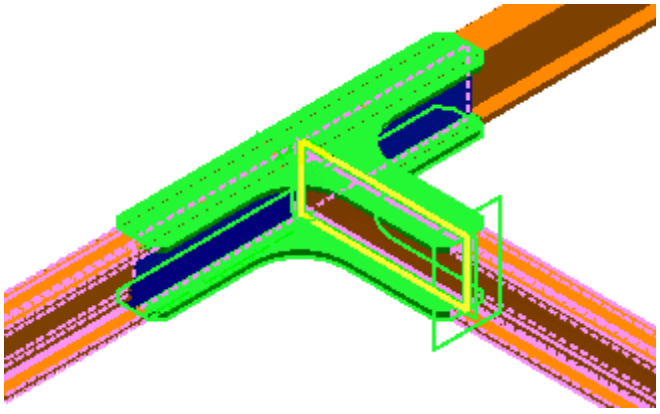


10. Select the supporting member to use as a reference for **Location**.



11. Select the location of the continuity plate in the **Location** list.
12. Click **Continue**.

    *A preview of the plate and the geometric constructions used to create the plate display in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



13. If the plate preview does not display on the correct side of the supported member, then:
    a. Click **Select Supports** .

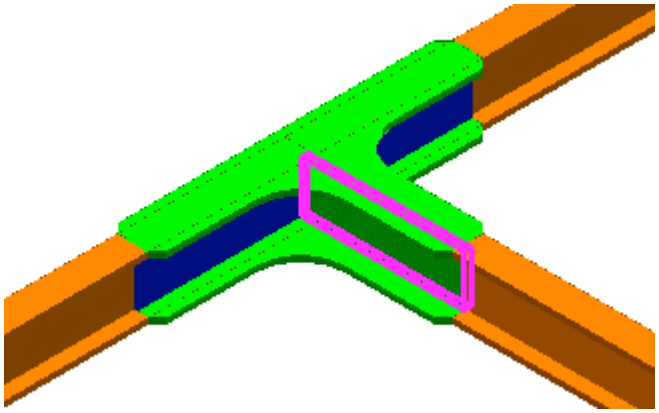
*The ribbon display returns to the **Continuity Plate** ribbon.*

b. Select another option from the **Location** list, click **Accept**, and then click **Continue**.

*A preview of the repositioned continuity plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

14. If needed, click **Define Boundaries**, and define boundaries for the plate.
15. Click **Finish**.

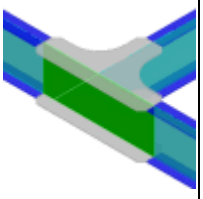
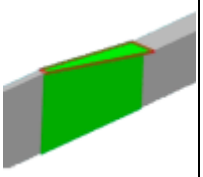
The software creates the continuity plate.



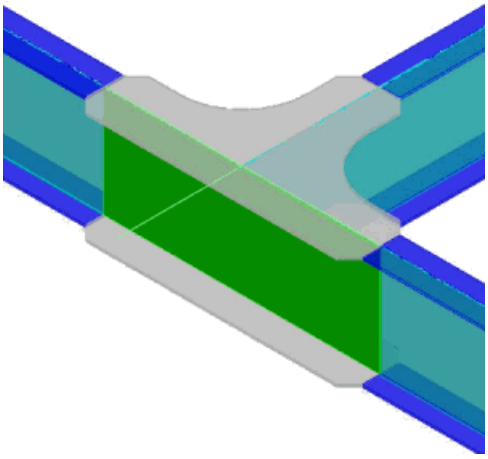
TIP If the macro does not properly cut through the member cross-sections, type a new distance to extend the trimming planes in the **Extension** box.

Transition Plates Tab (Advanced Plate Palette Dialog Box)

The following advanced plate system macros create transition plates between side or ring plates, and along the webs of supporting members. Transition plates are typically used to create offshore nodal connections between built-up members in combination with continuity plates, sides plates, and ring plates.

	**Transition Plate** - Creates a plate along supporting member webs between top and bottom side plates or ring plates. For more information, see *Transition Plate* (on page 329).
	**Gap Transition Plate** - Creates a plate to fill the gap between the webs or flanges of two profile systems or members of differing dimensions. For more information, see *Gap Transition Plate* (on page 334).

Transition Plate



Tooltip: TransitionPlate

Creates a plate used to transition along the webs of supporting members and between sets of side or ring plates. Transition plates are typically used to create offshore nodal connections between built-up members in combination with continuity plates, sides plates, and ring plates.

Transition Plate Ribbon

Properties

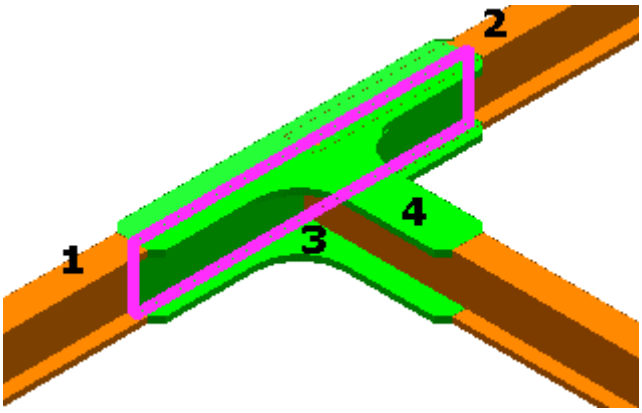
Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).

- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).



1 - Specifies the first member with which to establish the transition.

2 - Specifies the second opposing and collinear member with which to establish the transition.

3 - Specifies the first bounding side or ring plate.

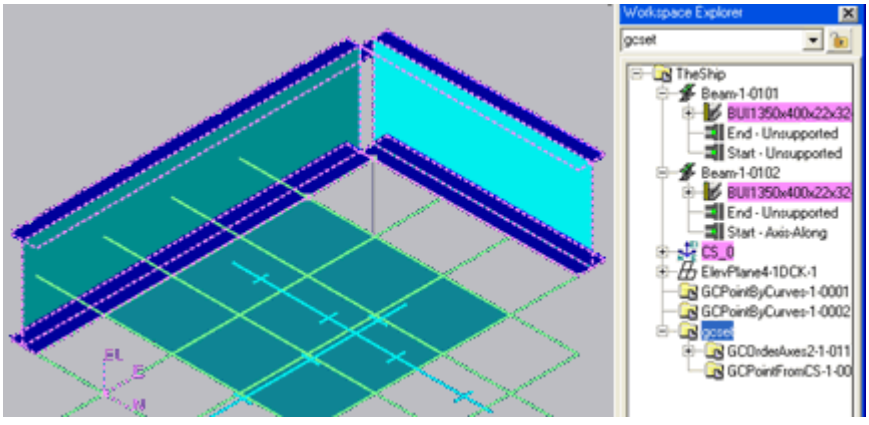
4 - Specifies the second bounding side or ring plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

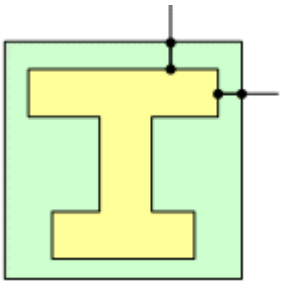
Accepts the selection for the current step.

Continue

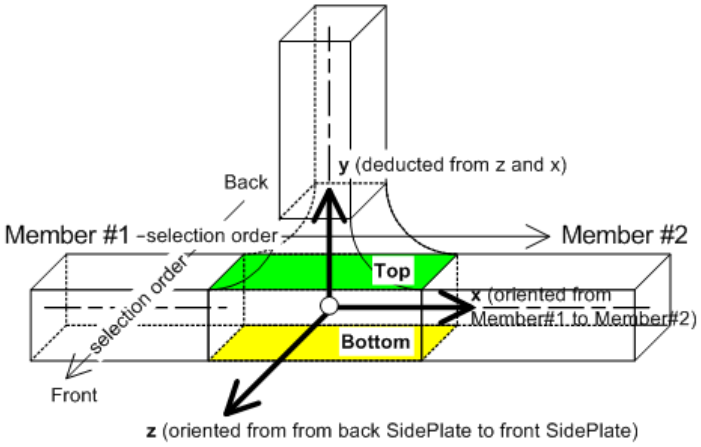
Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Extension - Specifies the distance to extend the trimming planes. If the macro does not properly extend the trimming planes through all of the member cross-section, you can increase this value.



Side - Specifies the location of the transition plate with respect to the computed coordinate system. Select **Bottom**, **Top**, or **Centered**.



What do you want to do?

- *Place a transition plate* (on page 332)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a transition plate

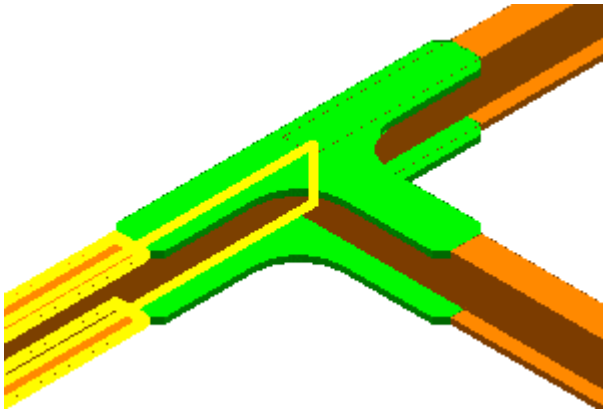
1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. At the nodal connection, create two ring plates or side plates.

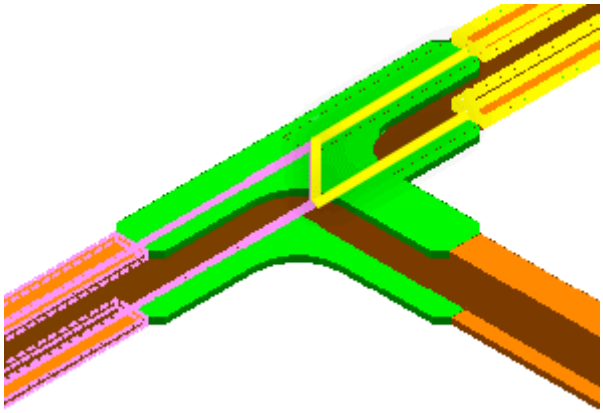
   Place a ring plate (on page 305)

   Place a side plate (on page 288)
4. On the **Transition Plates** tab, click **TransitionPlate**.

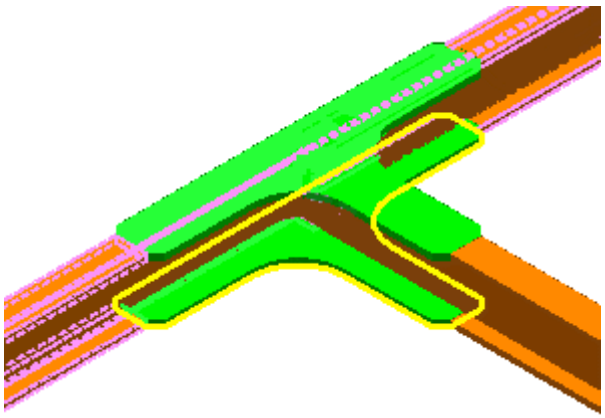
   *The **Transition Plate** ribbon displays.*
5. In a graphic view, select the first supporting member with which to establish the transition.



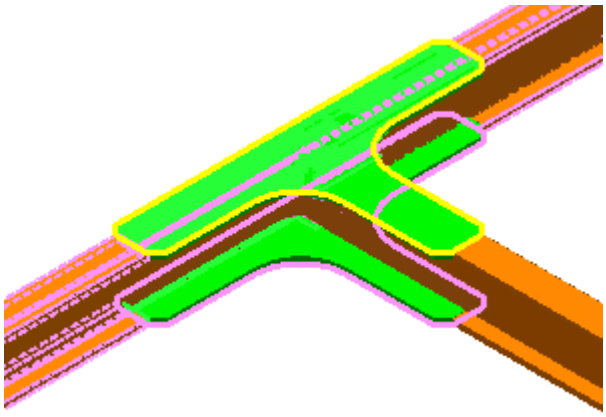
6. Select the second collinear member with which to establish the transition.



7. Select the first side or ring plate.

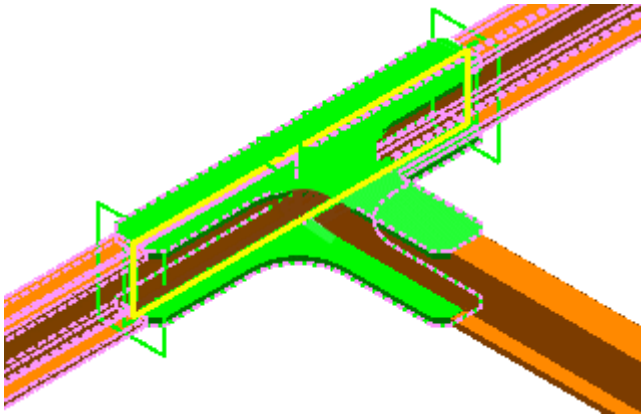


8. Select the second side or ring plate.



9. Select the location of the transition plate from the **Side** list.
10. Click **Continue**.

    *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



11. If the plate preview does not display on the correct side of the supporting members, then:
    a. Click **Select Supports** .

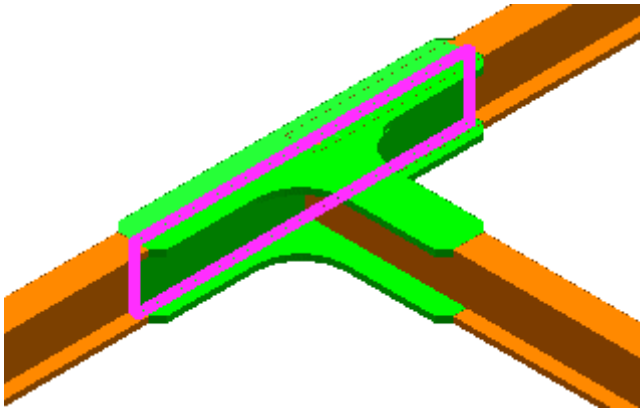
*The ribbon display returns to the **Transition Plate** ribbon.*

b. Select another option from the **Side** list, click **Accept**, and then click **Continue**.

*A preview of the repositioned transition plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

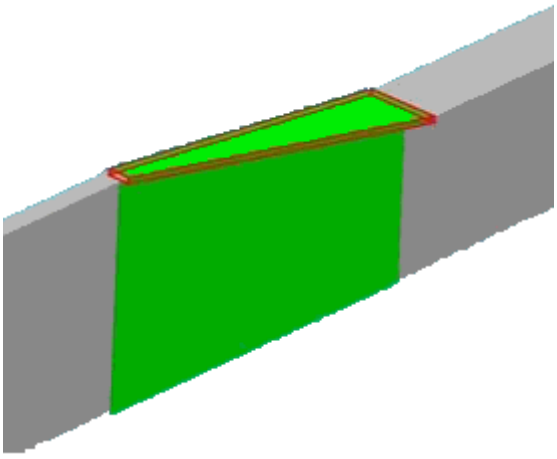
12. If needed, click **Define Boundaries**, and define boundaries for the plate.
13. Click **Finish**.

The software creates the transition plate.



TIP If the macro does not properly cut through the member cross-sections, type a new distance to extend the trimming planes in the **Extension** box.

Gap Transition Plate



Tooltip: TransitionPlateGap

Creates a plate to fill the gap between the webs or flanges of two profile systems or members of differing dimensions.

Gap Transition Plates Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the first member or profile system with which to establish the transition.

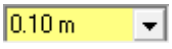
2 - Specifies the second member or profile system with which to establish the transition.

(3) - Optionally, select a bounding plate system. Typically, you select a plate system or a flange when creating a transition plate web.

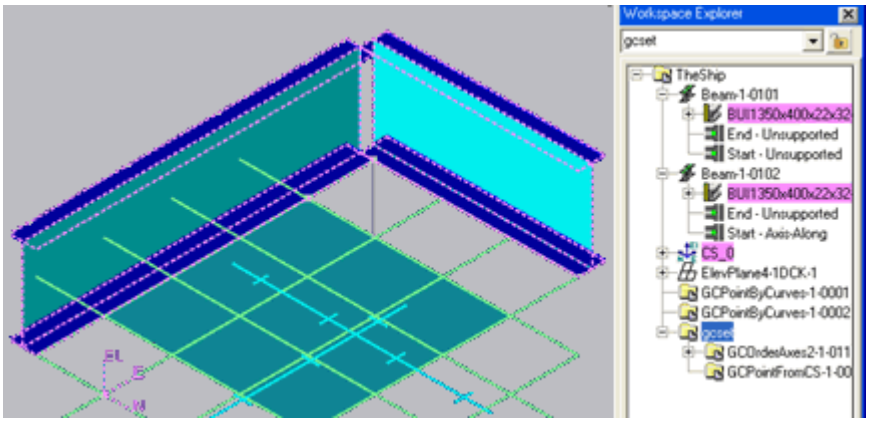
(4) - Optionally, select a second bounding plate system. Typically, you select a plate system or a flange when creating a transition plate web.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

Accepts the selection for the current step.

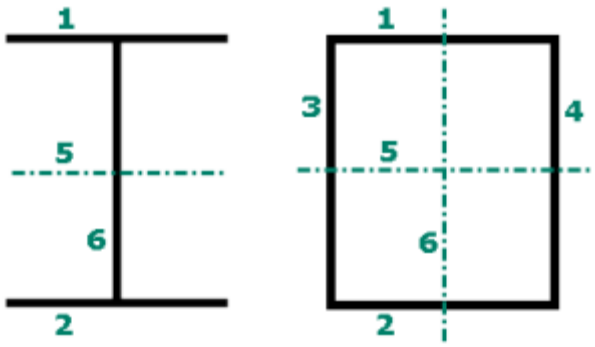
Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Category - Specifies a category for the macro. **This option is not currently used.**

Location - Specifies the location of the continuity plate with respect to the selected members. Select one of the following options:



1 - Top

2 - Bottom

3 - Left

4 - Right

5 - Centered-horiz

6 - Centered-vert

- **Top** - Transitions between the top flanges of profile systems or members, or between the bottom walls of box members.
- **Bottom** - Transitions between the bottom flanges of profile systems or members, or between the bottom walls of box members.

- **Left** - Transitions between the left side walls of box members.
- **Right** - Transitions between the right side walls of box members.
- **Centered-horiz** - Transitions between interior horizontal stiffeners of profile systems or members.
- **Centered-vert** - Transitions between the webs of profile systems or members, or between the interior horizontal stiffeners of box members.

What do you want to do?

- *Place a gap transition plate between flanges* (on page 337)
- *Place a gap transition plate between webs* (on page 339)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a gap transition plate between flanges

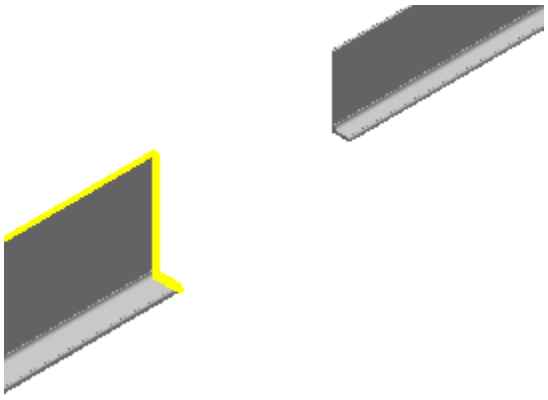
In this workflow, a gap transition plate is placed between flanges of two marine profile systems.

NOTE The parent plate system is hidden in the illustrations.

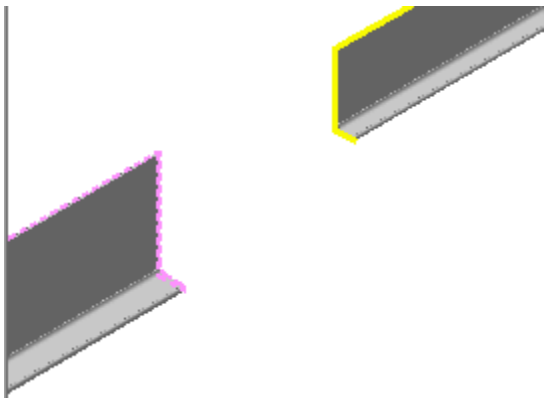
1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Transition Plates** tab, click **TransitionPlateGap**.

   *The **Gap Transition Plate** ribbon displays.*
4. In a graphic view, select the first profile system.

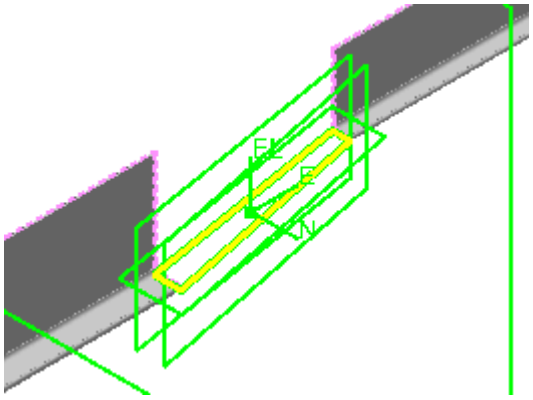


5. Select the second profile system.



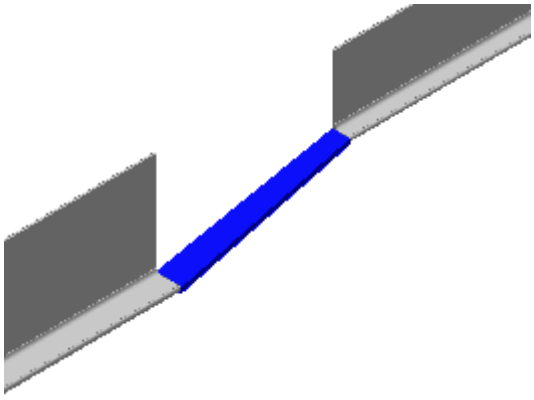
6. Select **Top** on the **Location** list.
7. Click **Continue**.

   *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



8. If needed, click **Define Boundaries** , and define boundaries for the plate.
9. Click **Finish**.

   The software creates the gap transition plate between the profile flanges.



Place a gap transition plate between webs

In this workflow, a gap transition plate is placed between webs of two marine profile systems.

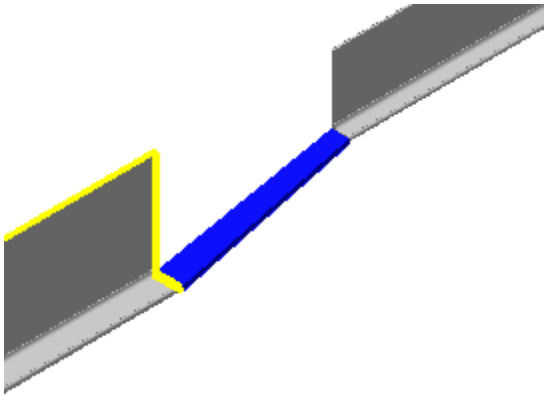
NOTE The parent plate system is hidden in the illustrations.

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

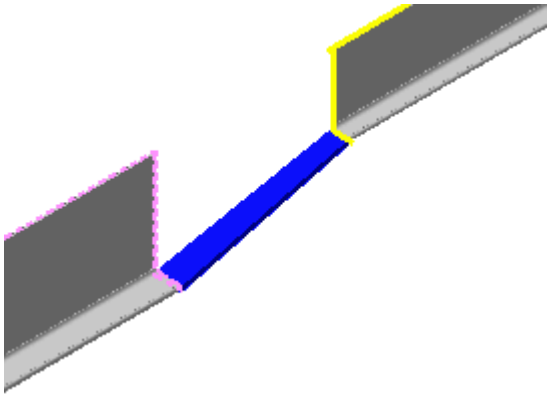
   *The **Advanced Plate Palette** dialog box displays.*
3. Create the gap transition plate between the profile flanges.

   Place a gap transition plate between flanges (on page 337)
4. On the **Transition Plates** tab, click **TransitionPlateGap**.

   *The **Gap Transition Plate** ribbon displays.*
5. In a graphic view, select the first profile system.

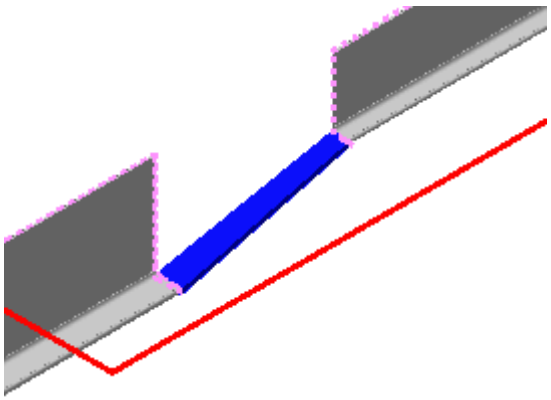


6. Select the second profile system.

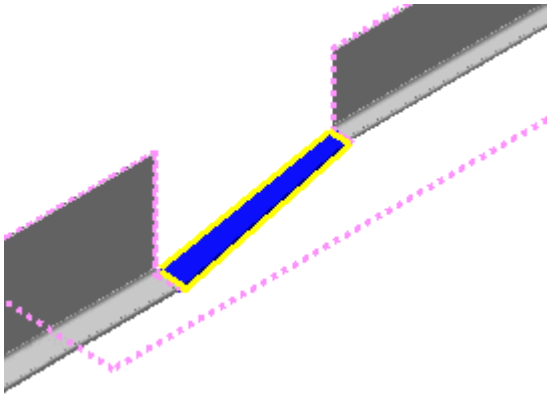


7. Click (3).

8. Select the parent plate system to bound the transition plate.

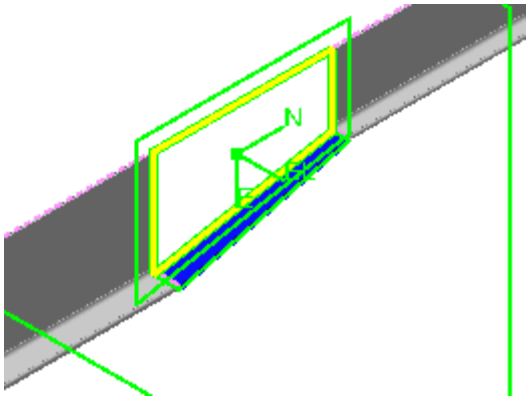


9. Click (4).
10. Select the flange transition plate to bound the opposite side of the transition plate.



11. Select **Centered-vert** on the **Location** list.
12. Click **Continue**.

    *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



13. If the plate preview does not display in the correct location, then:
    a. Click **Select Supports** .

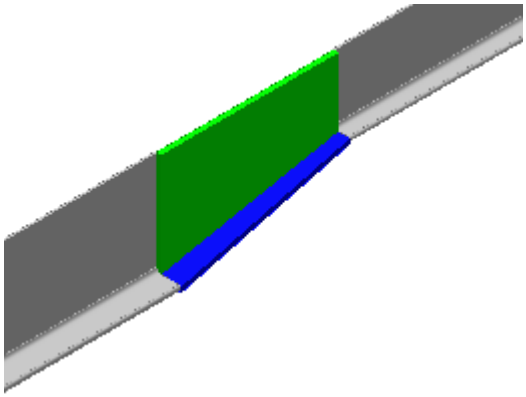
*The ribbon display returns to the **Gap Transition Plate** ribbon.*

b. Select another option from the **Location** list, click **Accept**, and then click **Continue**.

*A preview of the repositioned plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

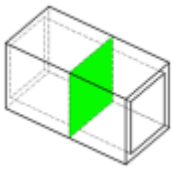
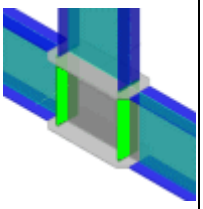
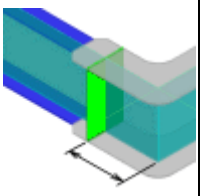
14. If needed, click **Define Boundaries**, and define boundaries for the plate.
15. Click **Finish**.

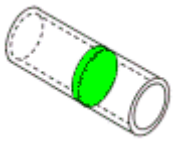
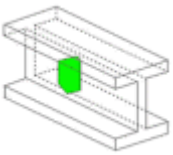
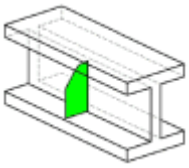
The software creates the gap transition plate between the profile webs.



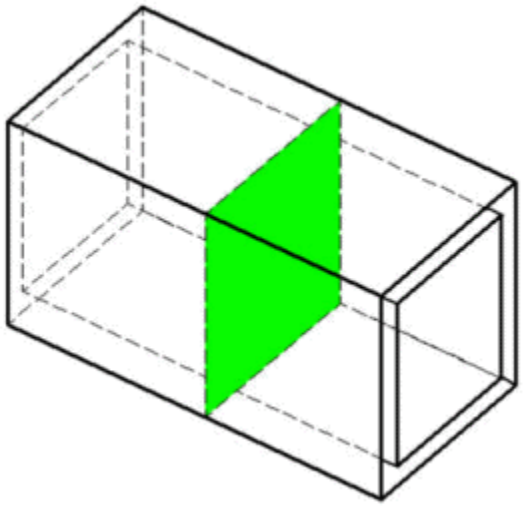
Buckling Plates Tab (Advanced Plate Palette Dialog Box)

The following advanced plate system macros create buckling plates on members or at nodal connections. A buckling plate stiffens the web and flange of a member. The buckling plate can be positioned by and bounded to plates sharing a nodal connection with the member, or can be defined by dimensions.

	**Boxed Member Buckling Plate** - Creates a buckling plate on the interior of a built-up boxed member. For more information, see *Boxed Member Buckling Plate* (on page 342).
	**Buckling Plate** - Creates a buckling plate aligned with one or two flanges of built-up members or with continuity plates. For more information, see *Buckling Plate* (on page 347).
	**Buckling Plate along Member** - Creates a buckling plate along a member at a nodal connection. For more information, see *Buckling Plate along Member* (on page 353).

	**Rolled Member Buckling Plate** - Creates a buckling plate at the intersection of the flanges of a rolled member with a nodal connection. For more information, see *Rolled Member Buckling Plate* (on page 358).
	**Tubular Member Buckling Plate** - Creates a buckling plate on the interior of a built-up tubular member. For more information, see *Tubular Member Buckling Plate* (on page 363).
	**Partial Buckling Plate** - Creates a partial buckling plate on one side of the web of a member and on one flange of the member. For more information, see *Partial Buckling Plate* (on page 368).
	**One-sided Buckling Plate** - Creates a buckling plate on one side of the web of a member. For more information, see *One-sided Buckling Plate* (on page 373).

Boxed Member Buckling Plate



Tooltip: Buckling_Box

Creates a buckling plate on the interior of a built-up boxed member. To locate the buckling plate, you specify the offset distance from the start node of the member.

Boxed Member Buckling Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

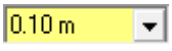
Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

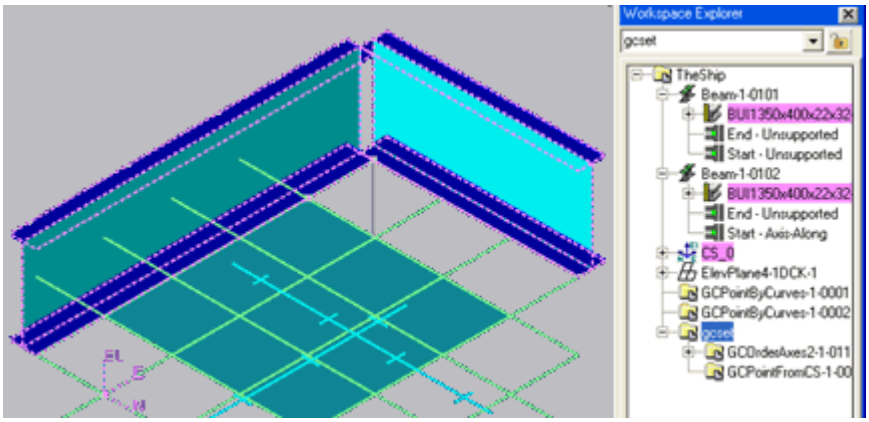
1 - Specifies the built-up boxed member.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

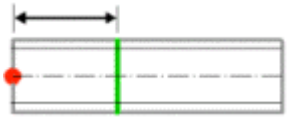
Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Distance - Specifies the location of the buckling plate as the distance from the start node of the member.



What do you want to do?

- *Place a buckling plate along a built-up boxed member* (on page 344)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a buckling plate along a built-up boxed member

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

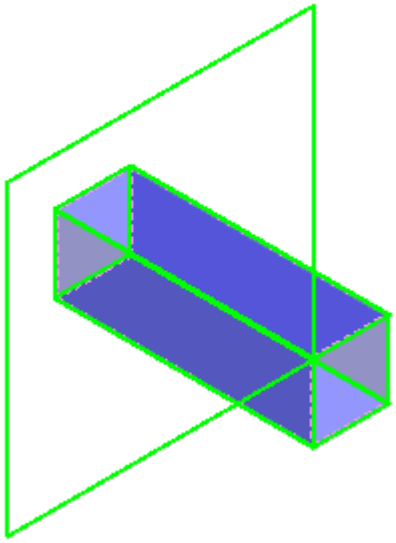
   *The **Advanced Plate Palette** dialog box displays.*

3. On the **Buckling Plate** tab, click **Buckling_Box**.

   *The **Boxed Member Buckling Plate** ribbon displays.*

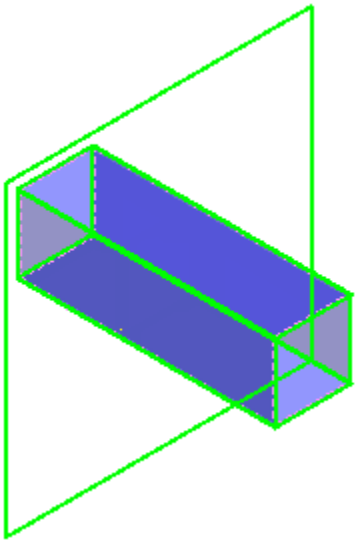
4. In a graphic view, select a built-up boxed member. The name of this member type typically starts with *BUBox...*.

   The intersection plane geometric construction, normal to the member axis and offset from the member start point, displays.



5. Type the required offset value for **Distance**, and then click **Accept** .

   The intersection plane moves to the new offset location.

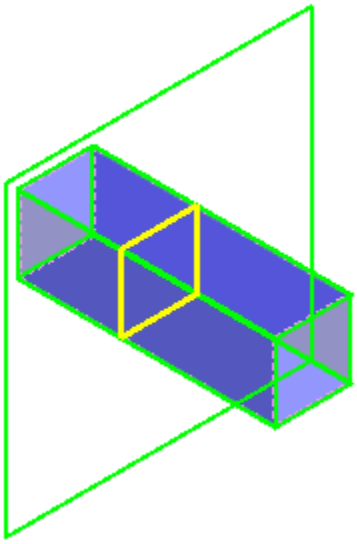


   TIP To better see the point at which the plane intersects the member, change the orientation of the graphic view.

6. Click **Continue**.

   A preview of the buckling plate displays in the graphic view. The ribbon display returns to the

Advanced Plate System *ribbon.*



7. If the plate preview does not display in the required location, then:

   a. Click **Select Supports** .

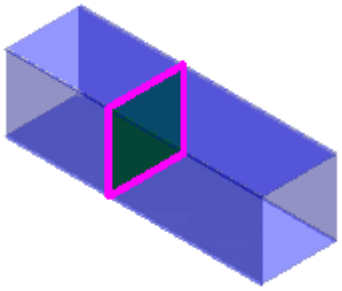
      The ribbon display returns to the ***Boxed Member Buckling Plate*** *ribbon.*

   b. Type a new offset value for **Distance**, click **Accept** , and then click **Continue**.

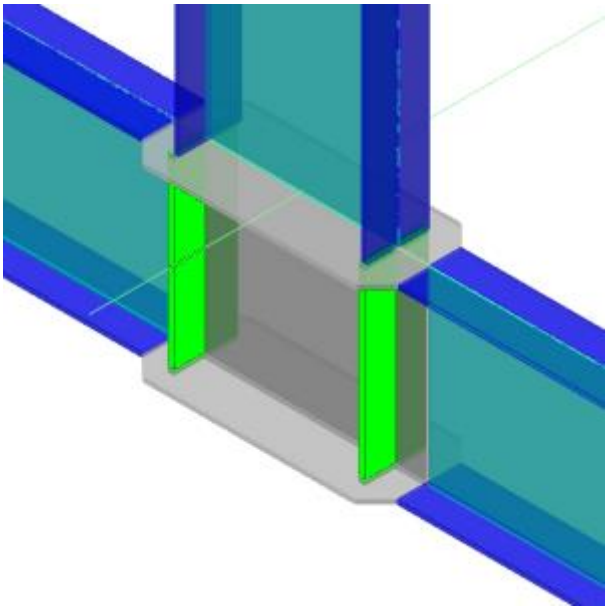
      A preview of the plate, now in the new offset location, displays in the graphic view. The ribbon display returns to the ***Advanced Plate System*** *ribbon.*

8. If needed, click **Define Boundaries** , and define boundaries for the plate.
9. Click **Finish**.

   The software creates the buckling plate.



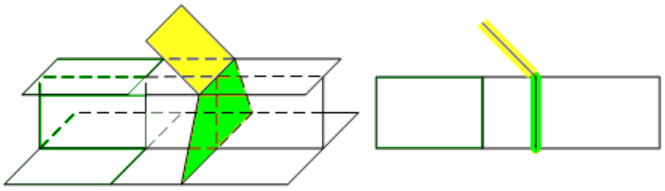
Buckling Plate



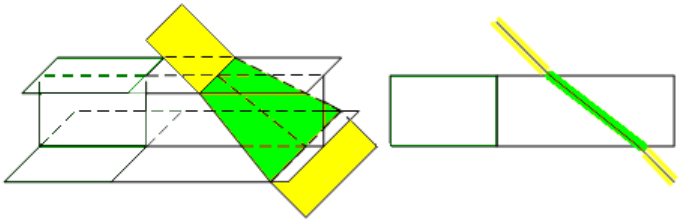
Tooltip:BucklingPlate

Creates a buckling plate aligned with one or two flanges of built-up members or with continuity plates. The buckling plate typically stiffens a transition plate and is limited (bounded) by two side plates.

If you specify one flange or continuity plate, then the buckling plate is positioned at the point at which the plate intersects the first side plate and orients normal to nodal connection plates. In the following example, the continuity plate is yellow, and the buckling plate is green.



If you specify two flanges or continuity plates, then the buckling plate is positioned between the intersection of the first flange and the first side plate, and the intersection of the second flange and the second side plate. In the following example, the continuity plates are yellow, and the buckling plate is green.



Buckling Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the first built-up flange or continuity plate with which to align the buckling plate.

2 - Specifies the first bounding plate, such as a side plate or a built-up flange.

3 - Specifies the second bounding plate, such as a side plate or a built-up flange.

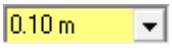
(4) - (Optional) Specifies a second built-up flange or continuity plate with which to align the buckling plate.

(5) - (Optional) Specifies a first lateral bounding plate, such as a transition plate.

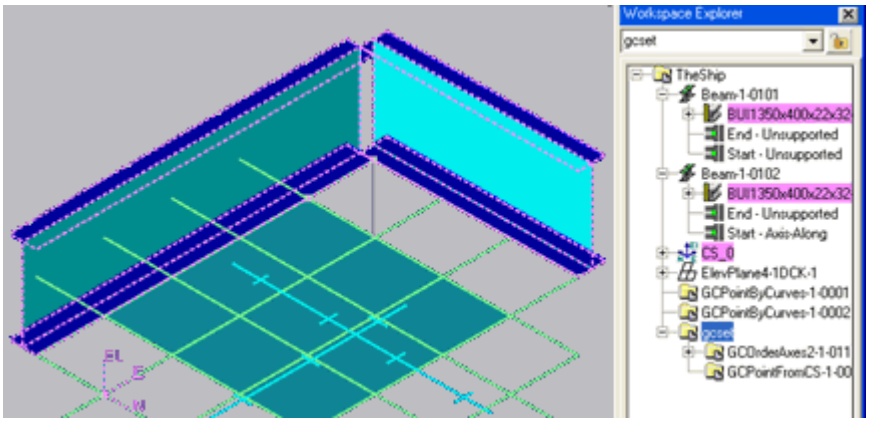
(6) - (Optional) Specifies a second lateral bounding plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

What do you want to do?

- *Place a buckling plate aligned with flanges or continuity plates* (on page 349)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

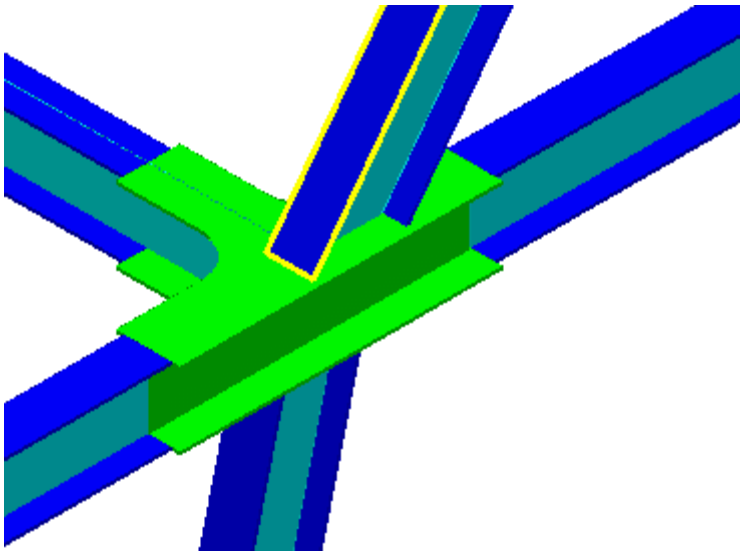
Place a buckling plate aligned with flanges or continuity plates

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

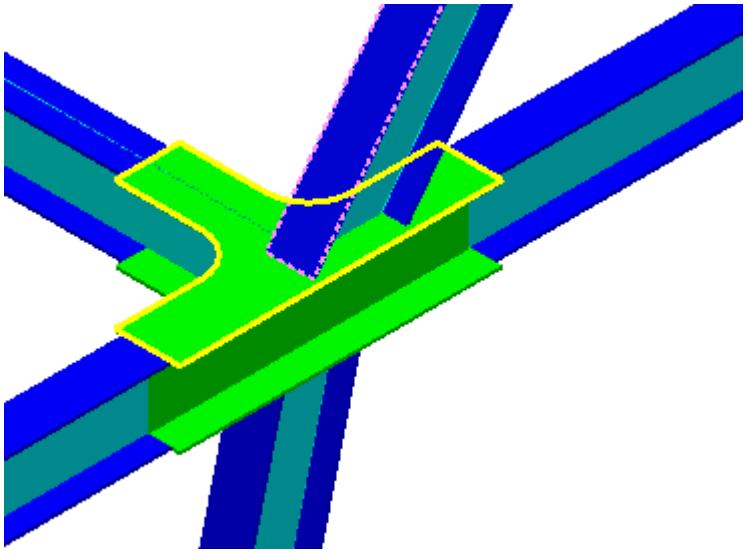
   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Buckling Plate** tab, click **BucklingPlate**.

   *The **Buckling Plate** ribbon displays.*

4. In a graphic view, select a built-up member flange or continuity plate incoming to the side or ring plate of a nodal connection. Flanges are used in the following example.

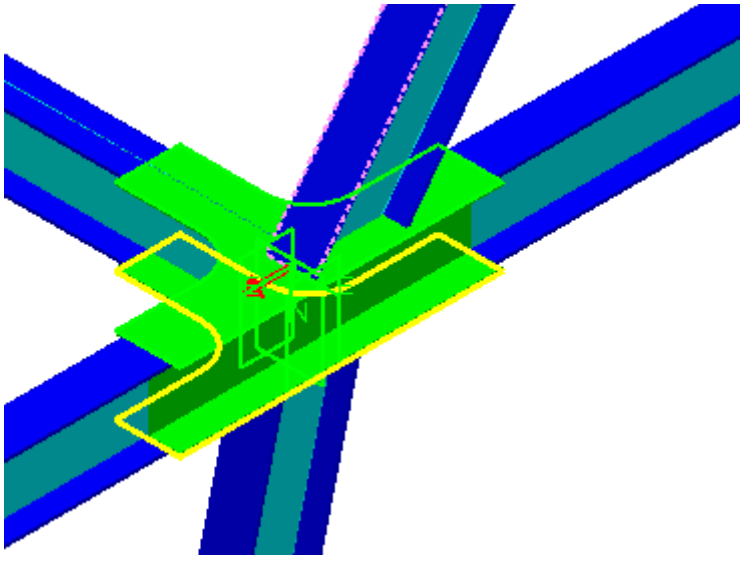


5. Select the first side or ring plate.



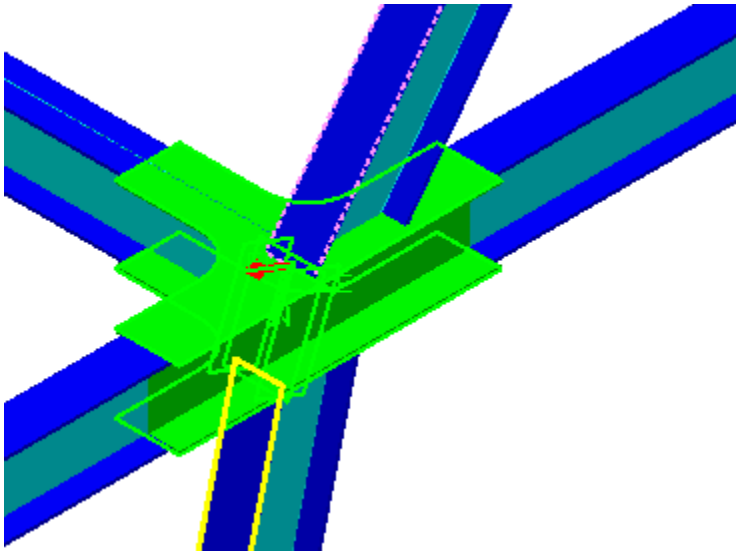
6. Select the second side or ring plate.

A preview of the geometric constructions used to create the plate displays.



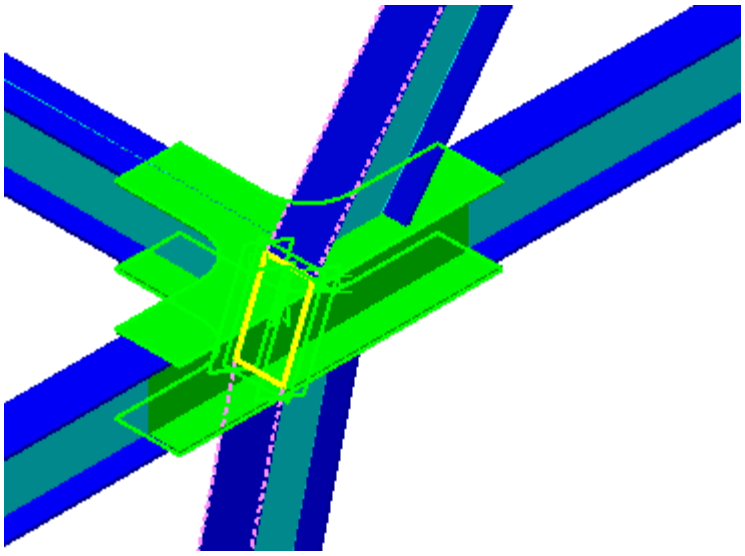
7. Optionally, click (4), and then select a second built-up member flange or continuity plate incoming to the side or ring plate.

   The geometric constructions adjust to transition between the two flanges.



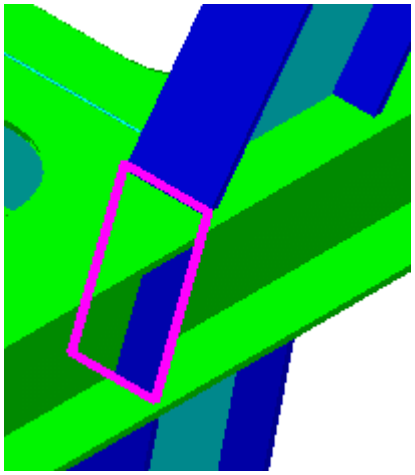
8. Click **Continue**.

A preview of the buckling plate displays in the graphic view. The ribbon display returns to the ***Advanced Plate System*** *ribbon.*

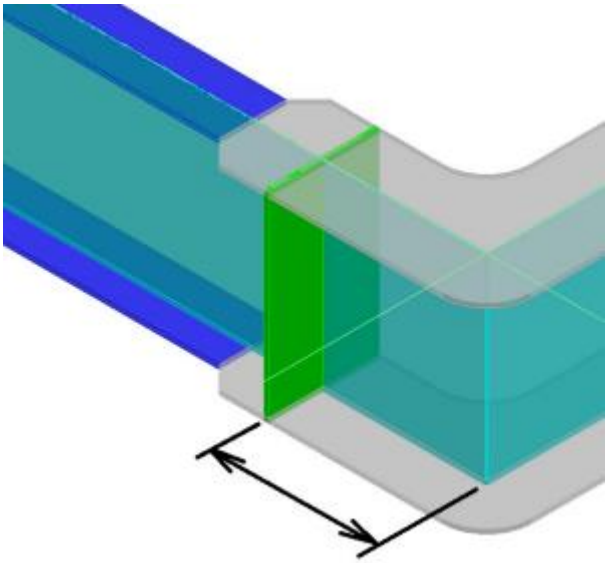


9. If needed, click **Define Boundaries** , and define boundaries for the plate.
10. Click **Finish**.

    The software creates the buckling plate.



Buckling Plate along Member



Tooltip: BucklingPlate0

Creates a buckling plate along a member at a nodal connection. To locate the buckling plate, you specify the offset distance from the node of the nodal connection.

Buckling Plate along Member Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the rolled or built-up member incoming to a nodal connection.

2 - Specifies the first bounding side or ring plate.

3 - Specifies the second bounding side or ring plate.

(4) - Optionally, select a first lateral bounding plate, such as a transition plate.

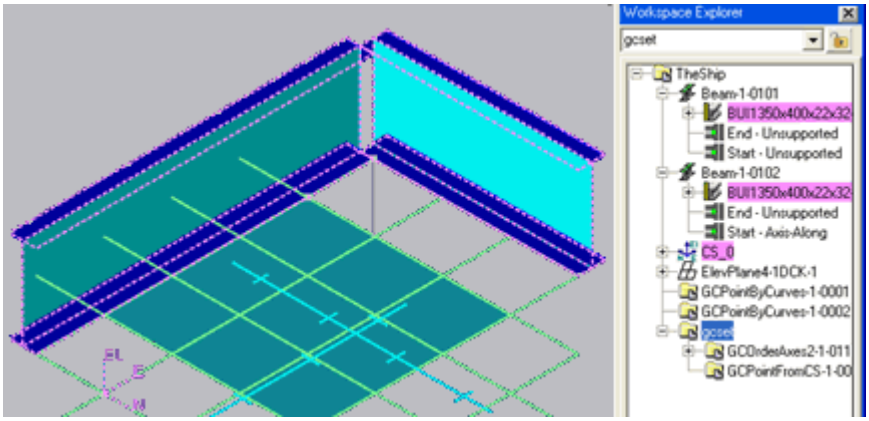
(5) - Optionally, select a second lateral bounding plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

- 1 - No background color when no input is selected.
- 1 - Yellow background when an input is newly selected.
- 0.10 m - Yellow background when you change a parameter from the default value.
- 1 - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

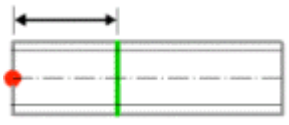
Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Distance from - Specifies the location of the buckling plate as the distance from the node of the nodal connection.



What do you want to do?

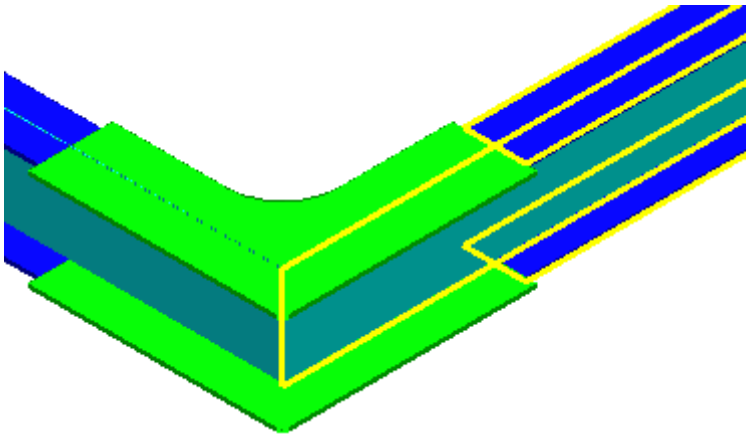
- *Place a buckling plate along a member* (on page 355)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a buckling plate along a member

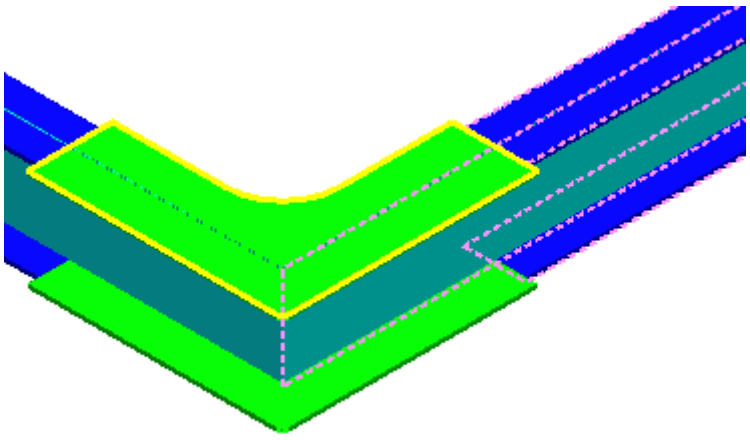
1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Buckling Plate** tab, click **BucklingPlate0**.

   *The **Buckling Plate along Member** ribbon displays.*
4. In a graphic view, select a rolled or built-up member.

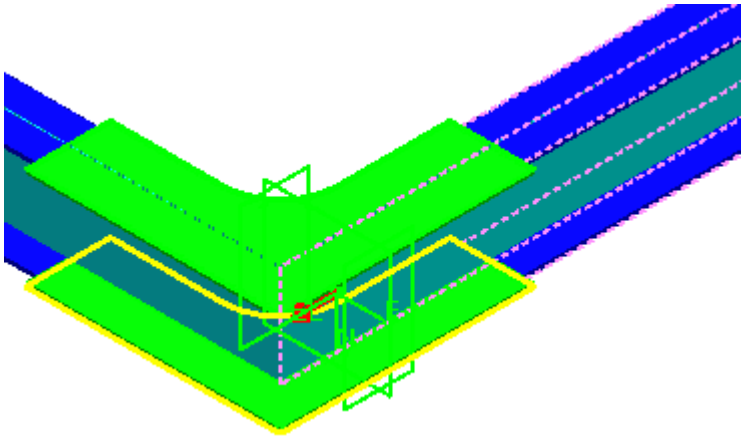


5. Select the first side or ring plate.



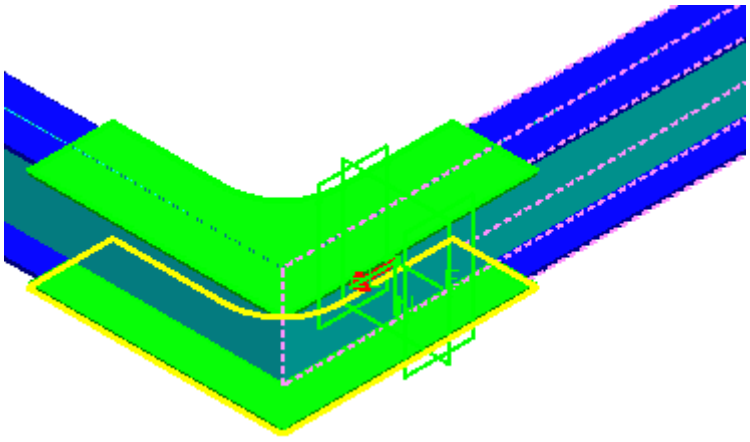
6. Select the second side or ring plate.

The geometric constructions used to create the plate display in the graphic view.



7. Type the required offset value for **Distance from**, and click **Accept** .

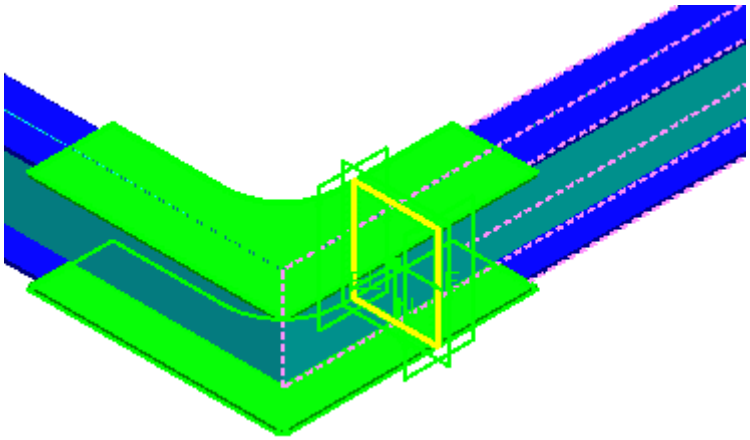
   The intersection plane moves to the new offset location.



   TIP To better see the point at which the plane intersects the member, change the orientation of the graphic view.

8. Click **Continue**.

*A preview of the buckling plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



9. If the plate preview does not display in the required location, then:
   a. Click **Select Supports** .

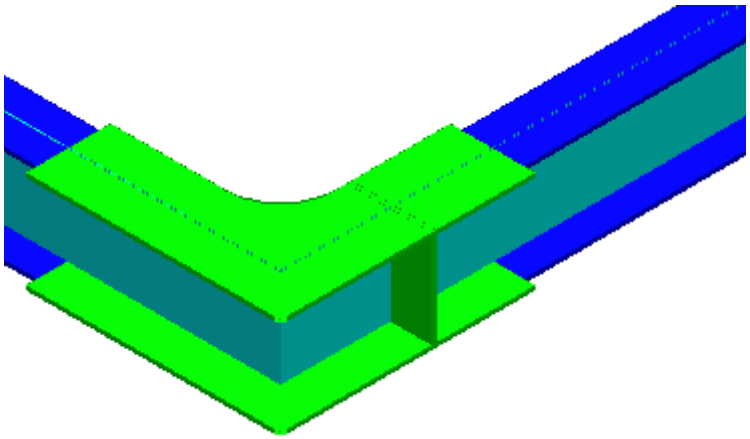
      *The ribbon display returns to the **Buckling Plate along Member** ribbon.*

   b. Type a new offset value for **Distance from**, click **Accept** , and then click **Continue**.

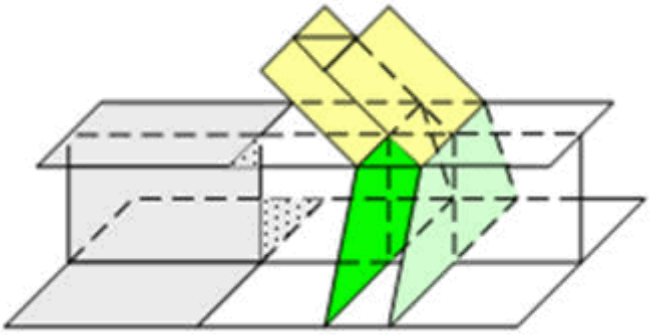
      *A preview of the plate, now in the new offset location, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

10. If needed, click **Define Boundaries** , and define boundaries for the plate.
11. Click **Finish**.

    The software creates the buckling plate.



Rolled Member Buckling Plate



Tooltip: BucklingPlate1r

Creates a buckling plate at the intersection of the flanges of a rolled member with a nodal connection. The buckling plate orients normal to nodal connection plates.

Rolled Member Buckling Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the rolled member incoming to the nodal connection.

2 - Specifies the first bounding plate, such as a side plate or an insert plate.

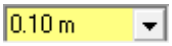
3 - Specifies the second bounding plate, such as a side plate or an insert plate.

(4) - (Optional) Specifies a first lateral bounding plate, such as a transition plate.

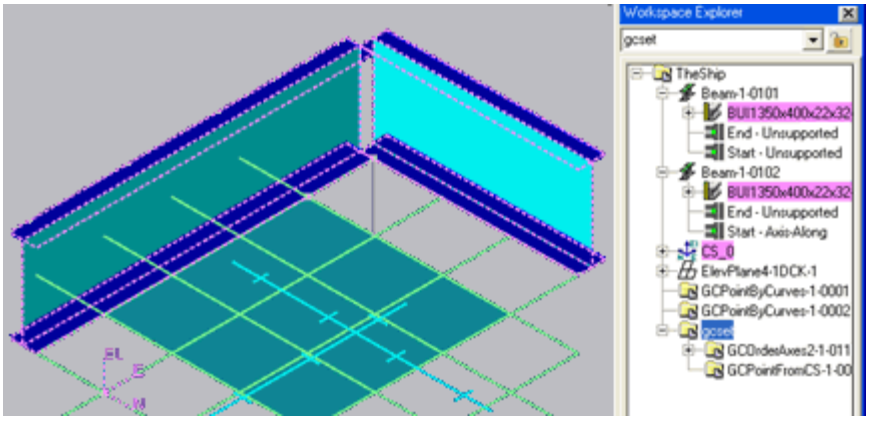
(5) - (Optional) Specifies a second lateral bounding plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

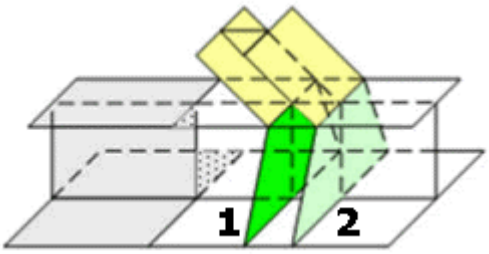
Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Location - Specifies the web of the member for placement of the buckling plate. Select **Inside** or **Outside**.



1 - Inside

2 - Outside

What do you want to do?

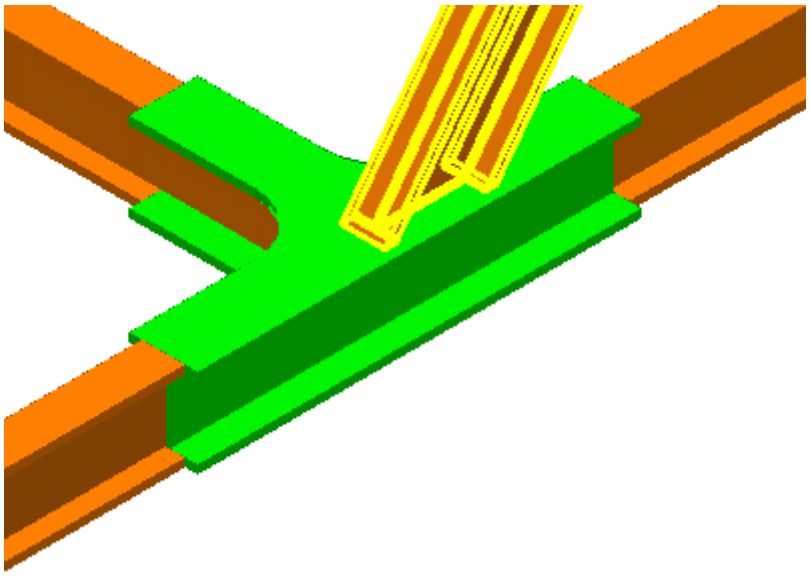
- *Place a buckling plate at a rolled member and a nodal connection* (on page 360)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a buckling plate at a rolled member and a nodal connection

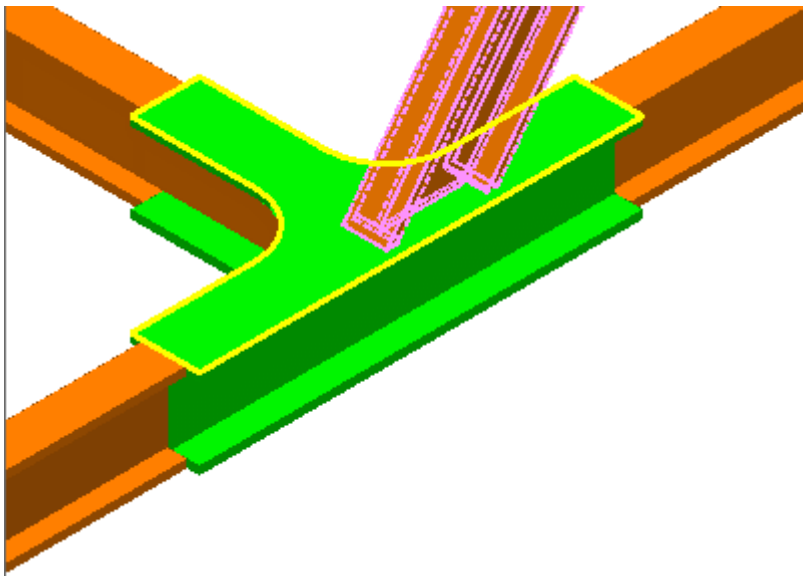
1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Buckling Plate** tab, click **BucklingPlate1r**.

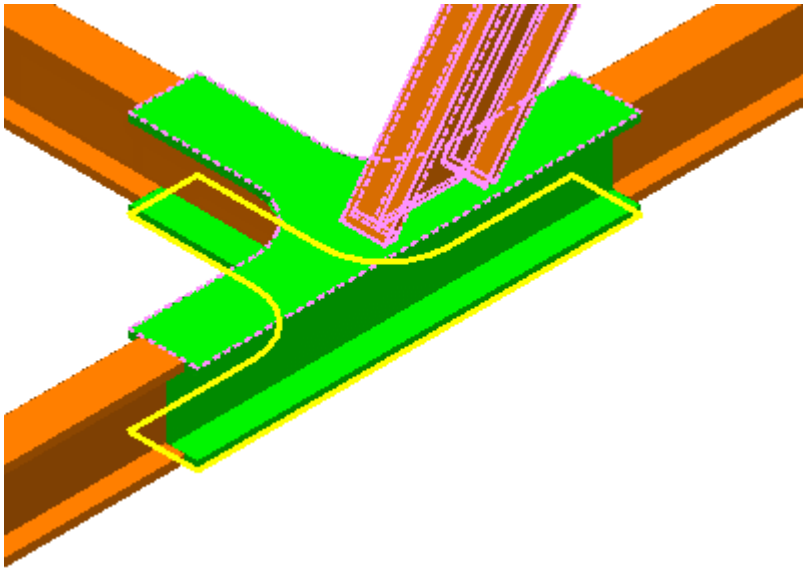
   *The **Rolled Member Buckling Plate** ribbon displays.*
4. In a graphic view, select a rolled member incoming to the side or ring plate of a nodal connection.



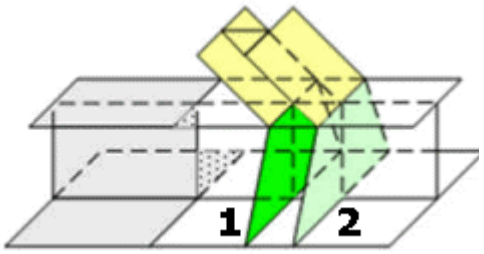
5. Select the first side or ring plate.



6. Select the second side or ring plate.



7. Select the required value for **Location**, and click **Accept** .

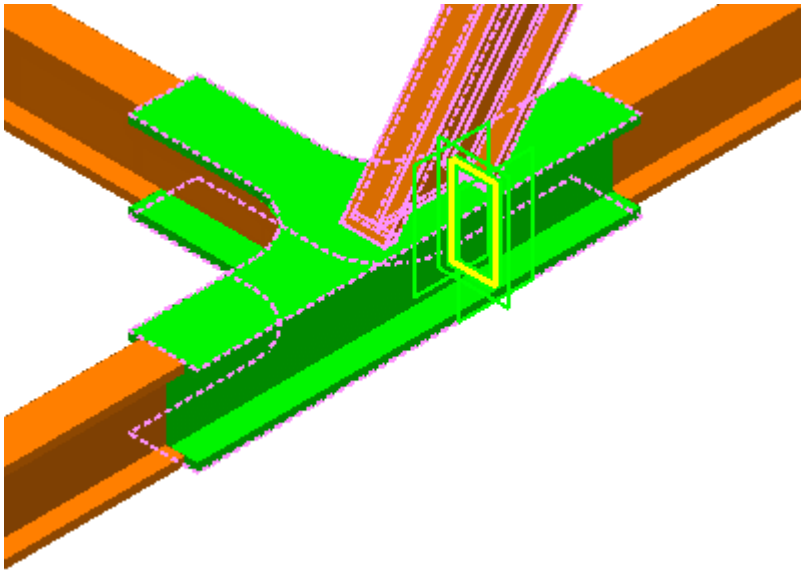


1 - Inside

2 - Outside

8. Click **Continue**.

   *A preview of the buckling plate and the geometric constructions used to create the plate display in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



9. If the plate preview does not display at the required flange, then:

   a. Click **Select Supports** .

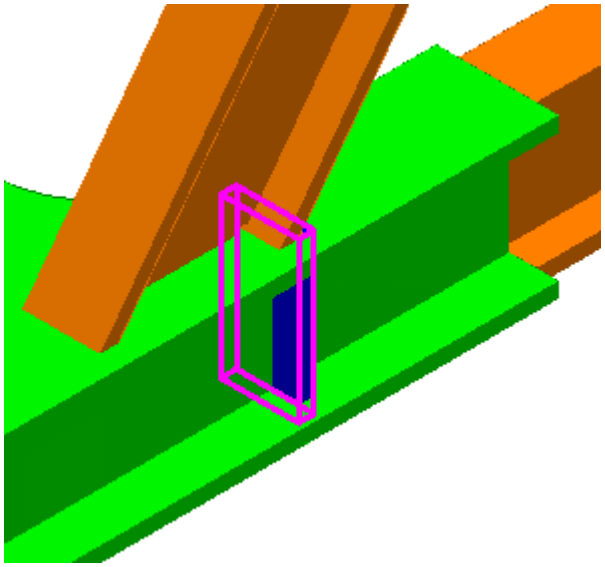
      *The ribbon display returns to the **Rolled Member Buckling Plate** ribbon.*

   b. Select a new value for **Location**, click **Accept** , and then click **Continue**.

      *A preview of the plate, now in the new location, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

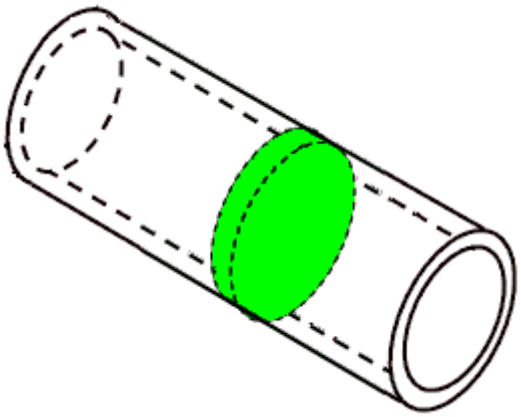
10. If needed, click **Define Boundaries** , and define boundaries for the plate.
11. Click **Finish**.

The software creates the buckling plate.



12. Repeat the previous steps to place a buckling plate at the other rolled member flange.

Tubular Member Buckling Plate



Tooltip: BucklingPlateForTube

Creates a buckling plate on the interior of a built-up tubular member. To locate the buckling plate, you specify the offset distance from the start or end node of the member.

Tubular Member Buckling Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the built-up tubular member.

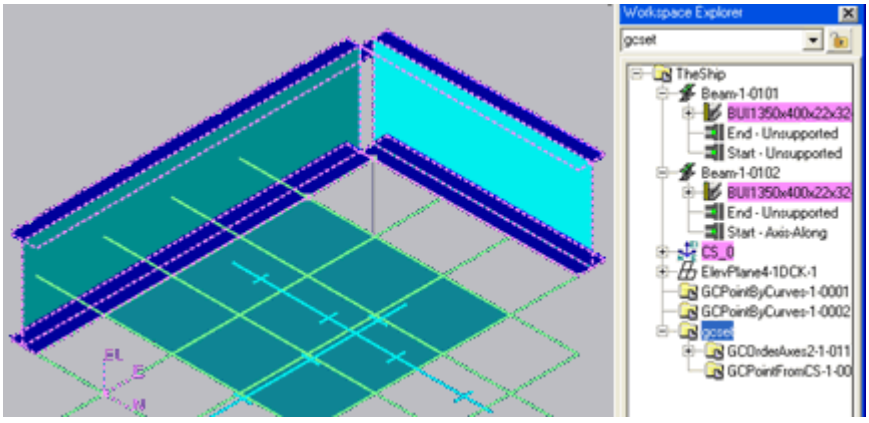
(2) - Select the member node used for offset distance measurement. The start node is the default selection.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

- 1 - No background color when no input is selected.
- 1 - Yellow background when an input is newly selected.
- 0.10 m - Yellow background when you change a parameter from the default value.
- 1 - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

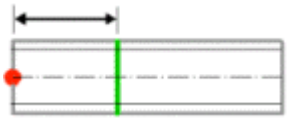
Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Distance - Specifies the location of the buckling plate as the distance from the selected node of the member.



What do you want to do?

- *Place a buckling plate along a built-up tubular member* (on page 365)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

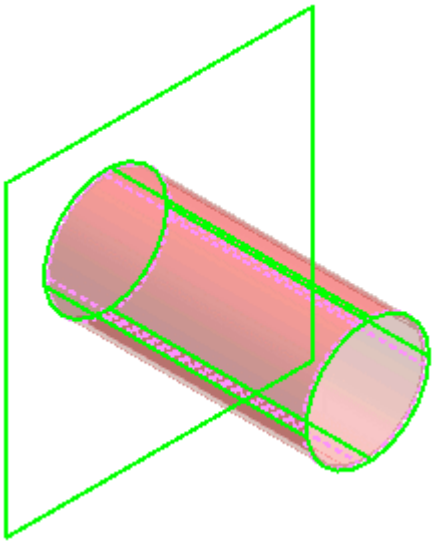
Place a buckling plate along a built-up tubular member

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Buckling Plate** tab, click **BucklingPlateForTube**.

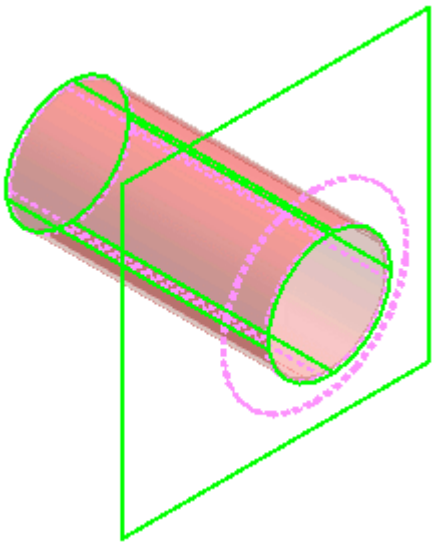
   *The **Tubular Member Buckling Plate** ribbon displays.*
4. In a graphic view, select a built-up tubular member. The name of this member type typically starts with *BUTube...*.

The intersection plane geometric construction, normal to the member axis and offset from the member start point, displays.



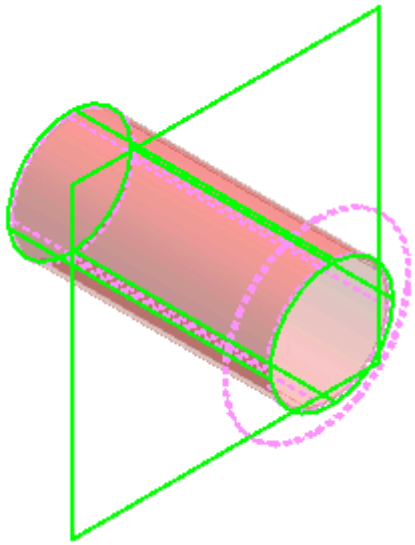
5. If a different node is needed, click (2), and then select the node.

   The new node highlights, and the intersection plane offsets from the new node.



6. Type the required offset value for **Distance**, and click **Accept** .

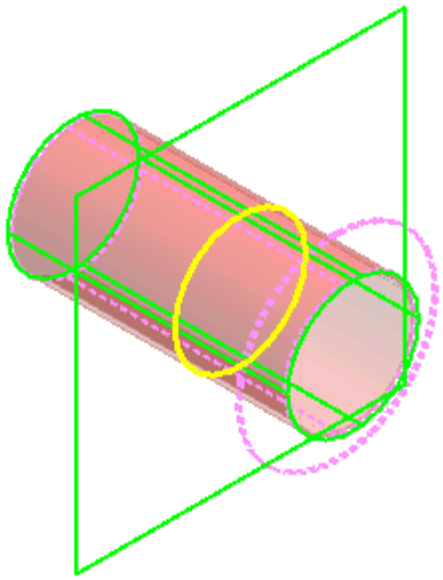
The intersection plane moves to the new offset location.



TIP To better see the point at which the plane intersects the member, change the orientation of the graphic view.

7. Click **Continue**.

   *A preview of the buckling plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



8. If the plate preview does not display in the required location, then:

   a. Click **Select Supports** .

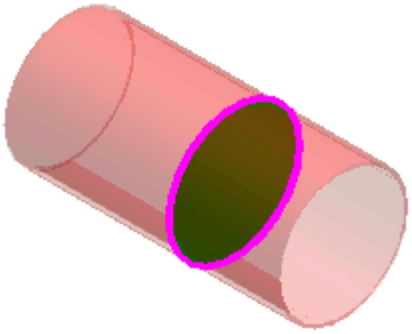
      *The ribbon display returns to the **Tubular Member Buckling Plate** ribbon.*

   b. Type a new offset value for **Distance**, click **Accept** , and then click **Continue**.

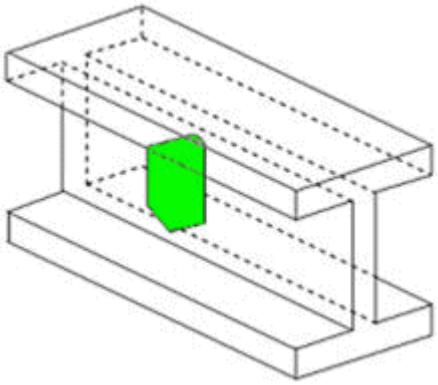
*A preview of the plate, now in the new offset location, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

9. If needed, click **Define Boundaries** , and define boundaries for the plate.
10. Click **Finish**.

    The software creates the buckling plate.



Partial Buckling Plate



Tooltip: BucklingPlatePartial

Creates a partial buckling plate on one side of the web of a member and on one flange of the member. To locate the buckling plate, you specify the offset distance from the start node of the member.

Partial Buckling Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

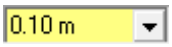
Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

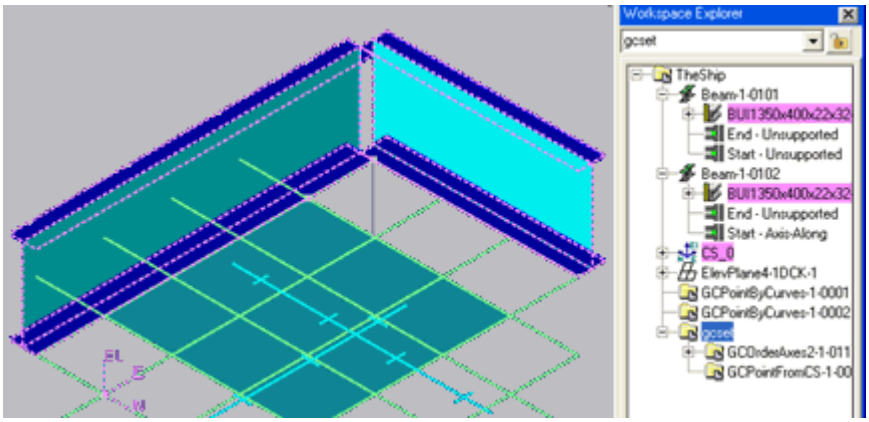
1 - Specifies the member. Select a rolled or a built-up member.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

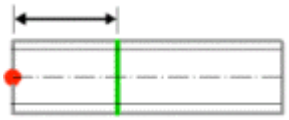
Accepts the selection for the current step.

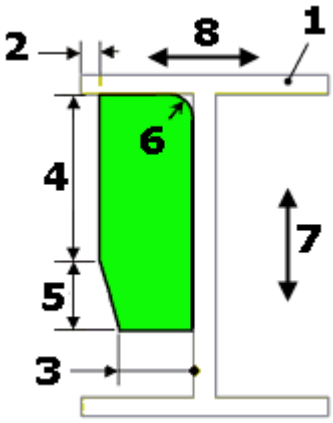
Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Offset - Specifies the location of the buckling plate as the distance from the start node of the member.





1 - Member

2 - Weld Toe 1

3 - Weld Toe 2

4 - Height 1

5 - Height 2

6 - Radius

7 - Top/Bottom

8 - Near/Far

Height 1 - Specifies the distance from the flange on the connected edge to the transition point.

Height 2 - Specifies the distance from the free edge of the buckling plate to the transition point.

NOTE **Height 1** plus **Height 2** is the total height of the partial buckling plate.

Weld Toe 1 - Specifies the offset distance between the edge of the member top flange and the buckling plate free edge at **Height 1**.

Weld Toe 2 - Specifies the offset distance between the edge of the member web and the buckling plate free edge at **Height 2**.

Radius - Specifies the interior corner radius.

Top/Bottom - Specifies the flange for placement of the connected edge. Select **Top** or **Bottom**.

Near/Far - Specifies the side of the member web for placement of the buckling plate. Select **Near** or **Far**.

What do you want to do?

- *Place a partial buckling plate on one side of a member web* (on page 371)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

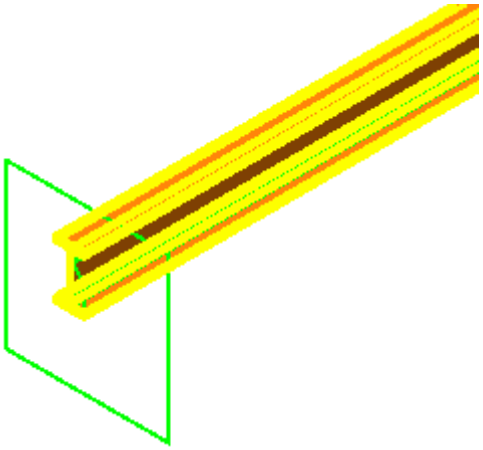
Place a partial buckling plate on one side of a member web

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Buckling Plate** tab, click **BucklingPlatePartial**.

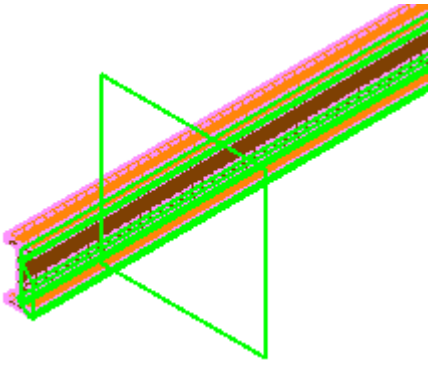
   *The **Partial Buckling Plate** ribbon displays.*
4. In a graphic view, select a member.

   The intersection plane geometric construction, normal to the member axis and offset from the member start point, displays.

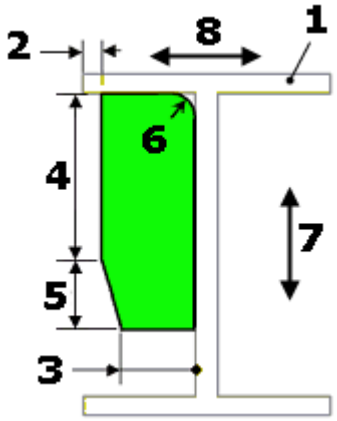


5. Type the required distance value for **Offset**, and press TAB.

   The intersection plane moves to the new offset location.



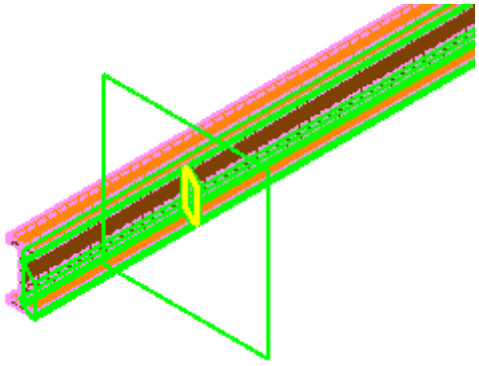
   TIP To better see the point at which the plane intersects the member, change the orientation of the graphic view.
6. Type the required values for **Height 1**, **Height 2**, **Weld Toe 1**, **Weld Toe 2**, and **Radius**.



1 - Member
2 - Weld Toe 1
3 - Weld Toe 2
4 - Height 1
5 - Height 2
6 - Radius
7 - Top/Bottom
8 - Near/Far

7. Select the required flange in the **Top/Bottom** box.
8. Select the required web side in the **Near/Far** box.
9. Click **Continue**.

   *A preview of the buckling plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



10. If the plate preview does not display in the required location, then:
    a. Click **Select Supports** .

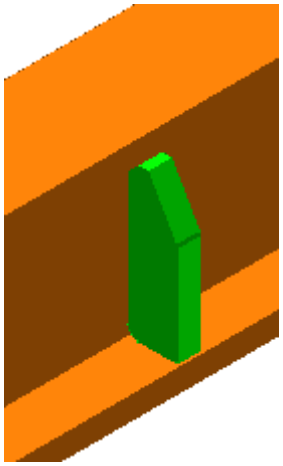
       *The ribbon display returns to the **Partial Buckling Plate** ribbon.*

    b. Select new values for **Near/Far** and **Top/Bottom**, click **Accept** , and then click **Continue**.

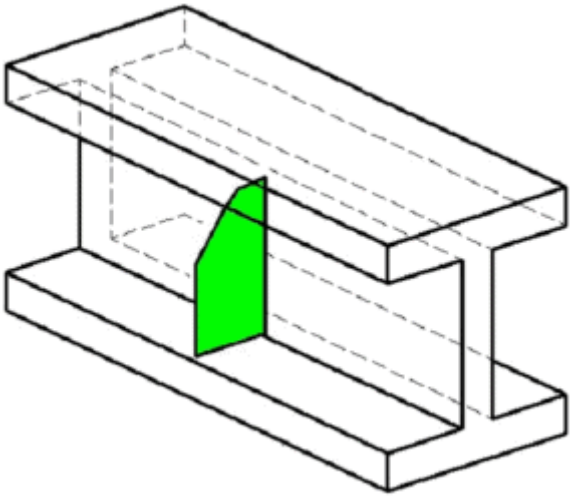
       *A preview of the plate, now in the new web side and flange side, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

11. If needed, click **Define Boundaries** , and define boundaries for the plate.
12. Click **Finish**.

The software creates the buckling plate.



One-sided Buckling Plate



Tooltip: BucklingPlateOneSide

Creates a buckling plate on one side of the web of a member. To locate the buckling plate, you specify the offset distance from the start node of the member.

One-sided Buckling Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

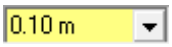
Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

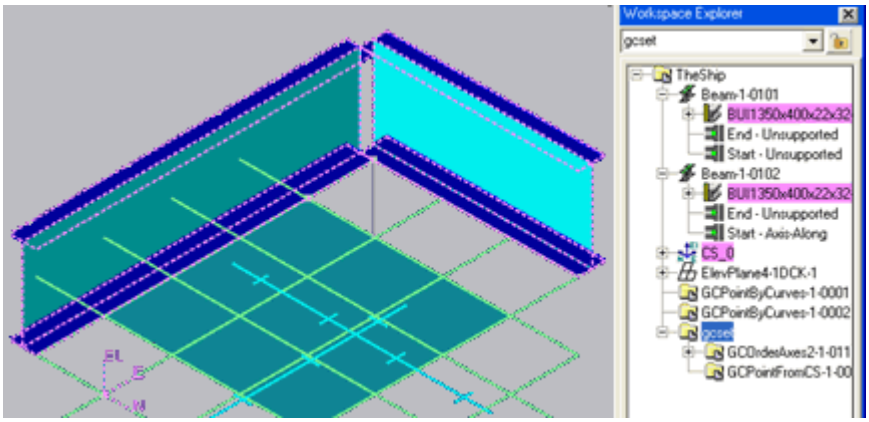
1 - Specifies the member. Select a rolled or a built-up member.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

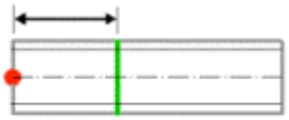
Accepts the selection for the current step.

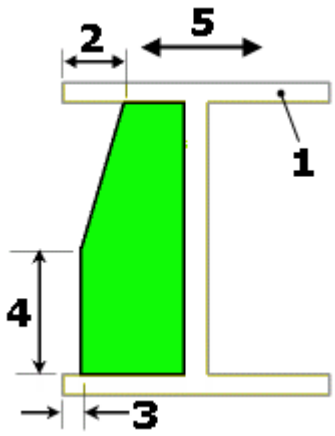
Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Offset - Specifies the location of the buckling plate as the distance from the start node of the member.





1 - Member

2 - Weld Toe 1

3 - Weld Toe 2

4 - Height

5 - Near/Far

Weld Toe 1 - Specifies the offset distance between the edge of the member top flange and the free edge of the buckling plate.

Weld Toe 2 - Specifies the offset distance between the edge of the member bottom flange and the free edge of the buckling plate.

Height - Specifies the distance from the bottom flange to the transition point. This option is applicable when the values of **Weld Toe 1** and **Weld Toe 2** differ.

Near/Far - Specifies the side of the member web for placement of the buckling plate. Select **Near** or **Far**.

What do you want to do?

- *Place a buckling plate on one side of a member web* (on page 375)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a buckling plate on one side of a member web

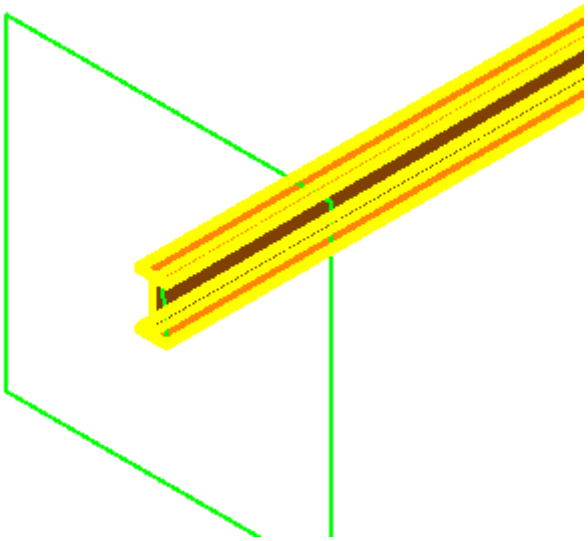
1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Buckling Plate** tab, click **BucklingPlateOneSide**.

*The **One-sided Buckling Plate** ribbon displays.*

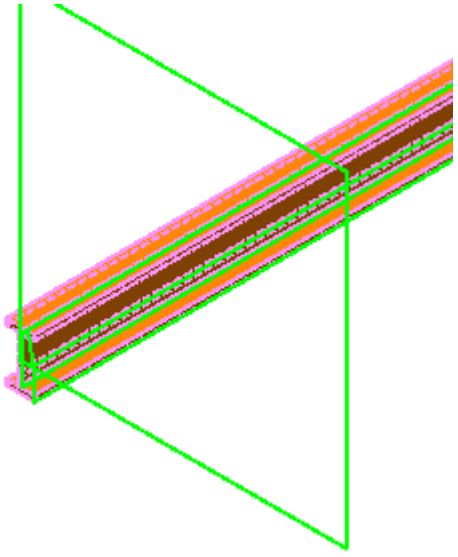
4. In a graphic view, select a member.

   The intersection plane geometric construction, normal to the member axis and offset from the member start point, displays.



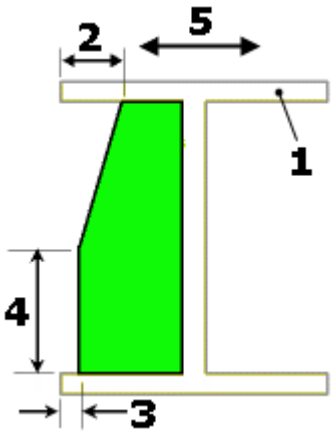
5. Type the required distance value for **Offset**, and press TAB.

   The intersection plane moves to the new offset location.



   TIP To better see the point at which the plane intersects the member, change the orientation of the graphic view.

6. Type the required values for **Weld Toe 1**, **Weld Toe 2**, and **Height**.



1 - Member

2 - Weld Toe 1

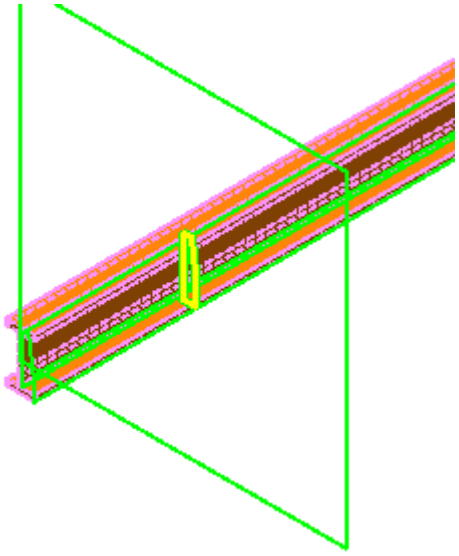
3 - Weld Toe 2

4 - Height

5 - Near/Far

7. Select the required web side in the **Near/Far** box.
8. Click **Continue**.

   *A preview of the buckling plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



9. If the plate preview does not display in the required location, then:
   a. Click **Select Supports** .

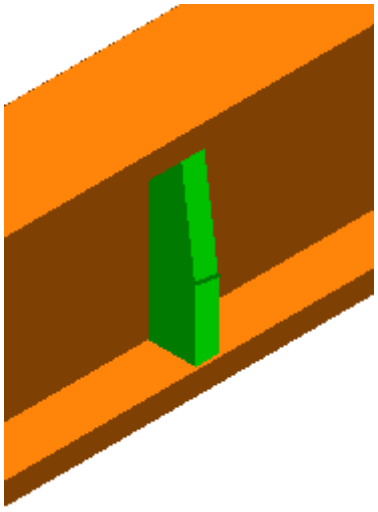
      *The ribbon display returns to the **One-sided Buckling Plate** ribbon.*

   b. Select a new value for **Near/Far**, click **Accept** , and then click **Continue**.

      *A preview of the plate, now in the new web side, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

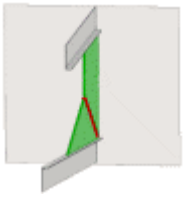
10. If needed, click **Define Boundaries** , and define boundaries for the plate.
11. Click **Finish**.

The software creates the buckling plate.

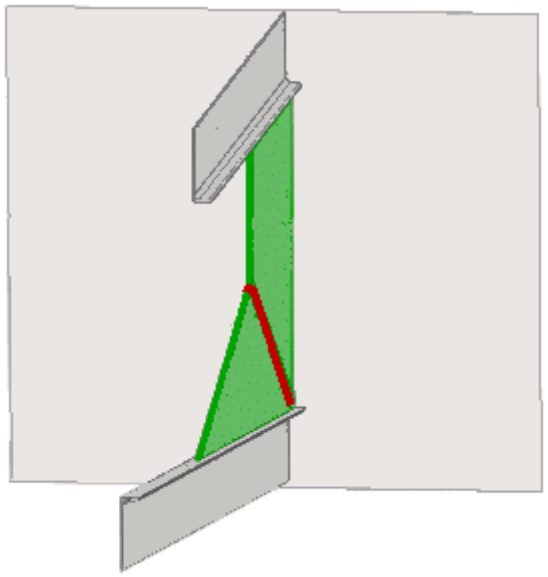


Brackets Tab (Advanced Plate Palette Dialog Box)

The following advanced plate system macro creates a bracket plate between supporting profile stiffeners systems and a supporting plate system.

	**Knuckled Bracket** - Creates a bracket plate containing a knuckle when the webs of supporting profiles are not coplanar. For more information, see *Knuckled Bracket* (on page 379).

Knuckled Bracket



Tooltip: KnuckledBracket

Creates a bracket plate containing a knuckle when the webs of supporting profiles are not coplanar. A bracket stiffens profile stiffeners and a supporting plate.

NOTE If the supports for a bracket are coplanar and a knuckled bracket is not needed, use **Bracket Plate System** . For more information, see *Bracket Plate System* (on page 227).

Knuckled Bracket Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the first profile system support. The widest side of the knuckle is along this system.

2 - Specifies the second profile system support.

3 - Specifies the plate system support.

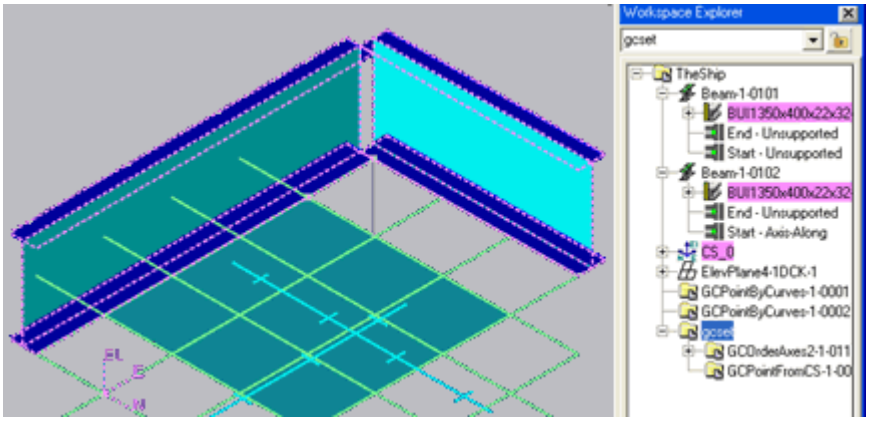
4 - Specifies the coordinate system. If there are multiple coordinate systems, select the coordinate system to use for justification of the bracket.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

- 1 - No background color when no input is selected.
- 1 - Yellow background when an input is newly selected.
- 0.10 m - Yellow background when you change a parameter from the default value.
- 1 - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

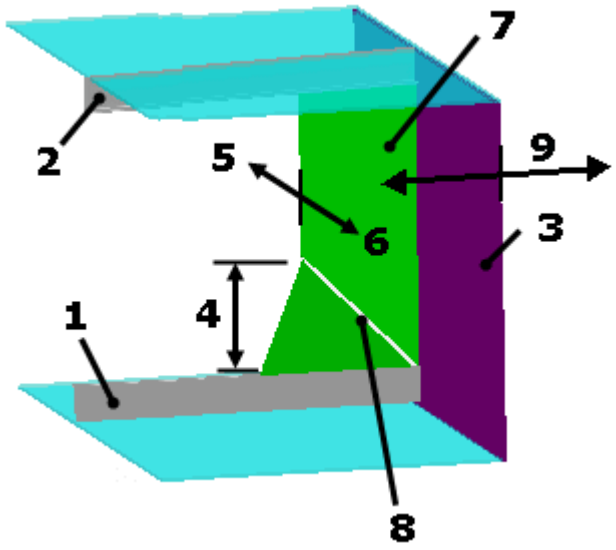
Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.



1 - First profile system

2 - Second profile system

3 - Plate system

4 - Knuckle height for partial knuckle

5 - Left for **Justif #1** and **Justif #2**, and direction of negative value for **U-offset #1** and **U-offset #2**.

6 - Right for **Justif #1** and **Justif #2**, and direction of positive value for **U-offset #1** and **U-offset #2**.

7 - Bracket

8 - Bracket knuckle

9 - Positive/negative directions for **Orientation**.

Auto Preview - Specifies whether a preview of the bracket displays. The preview updates as options are changed. Select **On** or **Off**.

Orientation - Specifies the direction of the bracket relative to the selected plate system **3** and the selected coordinate system **4**.

Width - Specifies the width of the bracket as measured from the plate system.

Knuckle Type - Specifies whether the knuckle extends to the full height of the plate. Select **Partial** to define a height of the knuckle from profile system **1**. Select **Full t**o define the height of the knuckle the full height between profile systems **1** and **2.**

Knuckle Height - Defines the height of the knuckle from profile system **1**. This option is only available when **Full** is selected for **Knuckle Type**.

Justif #1 - Specifies the justification of the bracket on the flange of profile system **1**. Select **Centered** to center the bracket on the flange. Select **Left** to position the bracket on the left side of the flange. Select **Right** to position the bracket on the right side of the flange. **Left** and **Right** are defined by looking toward the free edge of the bracket when the top of your view is toward support and are independent of the secondary orientation of the profile system supports. The default value is **Centered**.

U-offset #1 - Specifies the distance to offset the bracket on the flange of profile system **1**. A positive value offsets the bracket to the right when looking toward the free edge of the bracket. A negative value offsets the bracket to the left when looking toward the free edge of the bracket. The maximum value is one-half of the flange width.

Justif #2 - Specifies the justification of the bracket on the flange of profile system **2**. Select **Centered** to center the bracket on the flange. Select **Left** to position the bracket on the left side of the flange. Select **Right** to position the bracket on the right side of the flange. **Left** and **Right** are defined by looking toward the free edge of the bracket and are independent of the secondary orientation of the profile system. The default value is **Centered**.

U-offset #2 - Specifies the distance to offset the bracket on the flange of profile system **2**. A positive value offsets the bracket to the right when looking toward the free edge of the bracket. A negative value offsets the bracket to the left when looking toward the free edge of the bracket. The maximum value is one-half of the flange width.

What do you want to do?

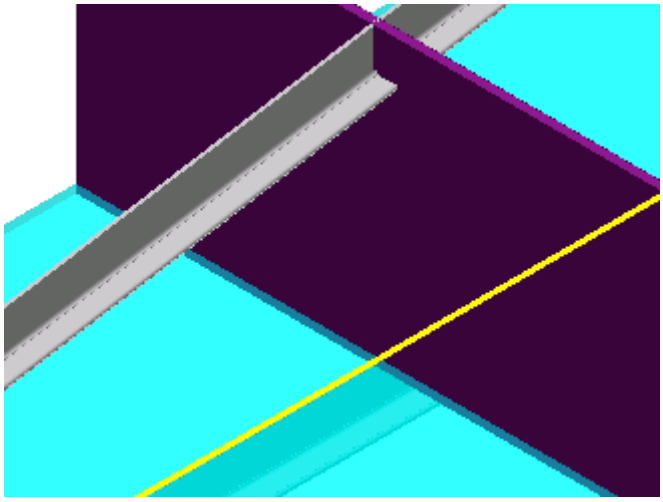
- *Place a knuckled bracket* (on page 382)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a knuckled bracket

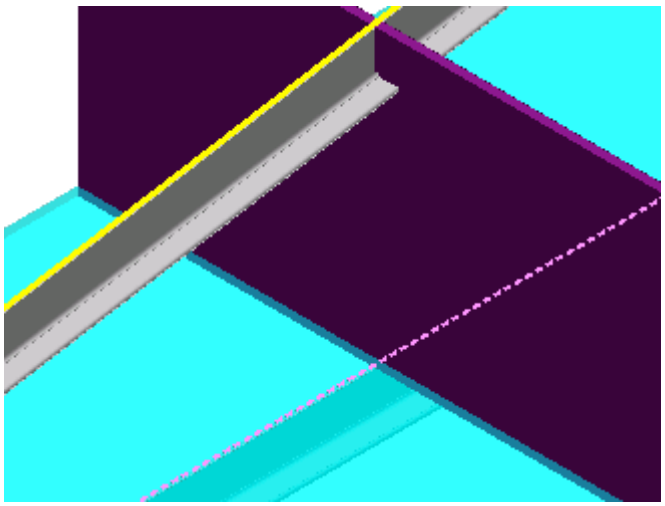
1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Brackets** tab, click **KnuckledBracket**.

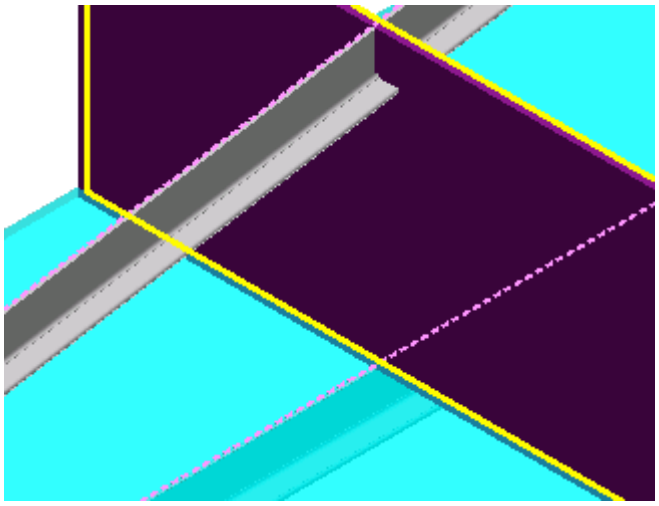
   *The **Knuckled Bracket** ribbon displays.*
4. In a graphic view, select the first profile system support. The widest side of the knuckle is along this system.



5. Select the second profile system support.

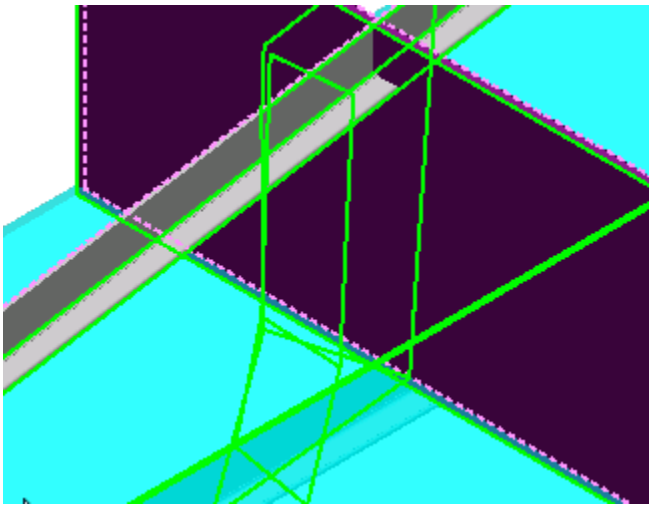


6. Select the plate system support.



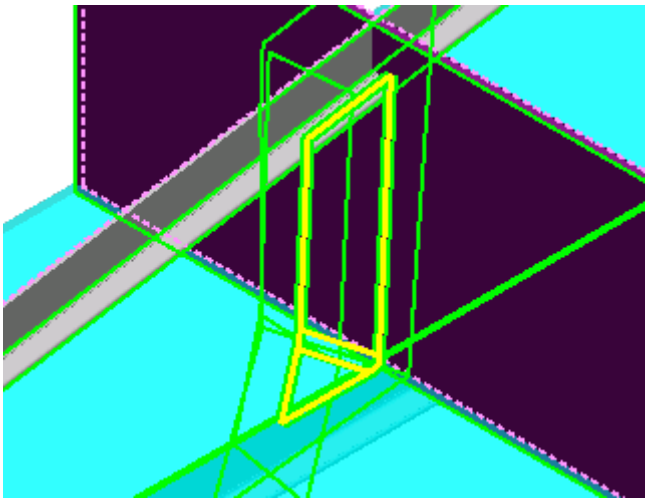
7. Select the coordinate system.

A preview of the geometric constructions used to create the bracket displays in the graphic view.



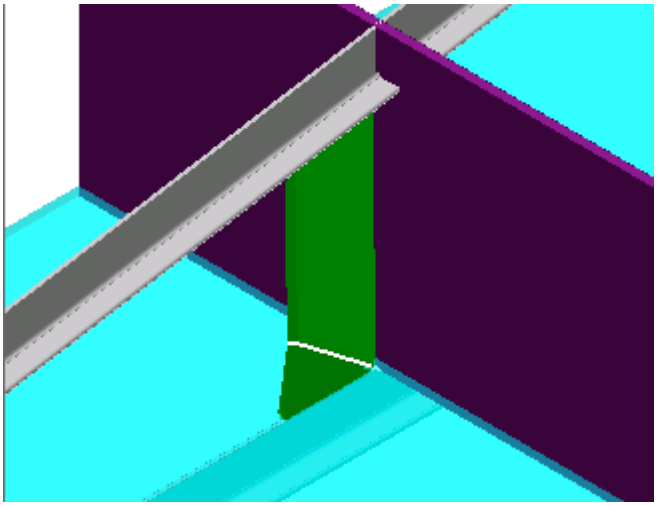
8. If the bracket is not on the required side of the plate system, select an axis in the opposite direction in the **Orientation** box. For example, in the previous figure, if **-x** is the value, then select **+x** to switch the bracket to the other side of the plate.
9. Type the required value for **Width**.
10. Select the knuckle type. If it is a partial knuckle, type a value for **Knuckle Height**.
11. Set values for justification (**Justif #1** and **Justif #2**) and offset (**U-offset #1** and **U-offset #2**) at each end of the bracket.
12. Click **Continue**.

    *A preview of the bracket displays in the graphic view, and the **Advanced Plate System** ribbon returns.*



    NOTE If changes to the bracket parameters are needed, click **Select Supports** to return to the **Knuckled Bracket** ribbon, and repeat the previous steps.
13. Click **Finish**.

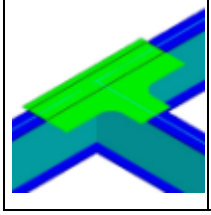
The software creates the knuckled bracket.

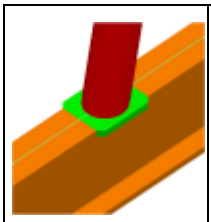
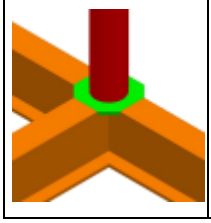


Insert Plate Tab (Advanced Plate Palette Dialog Box)

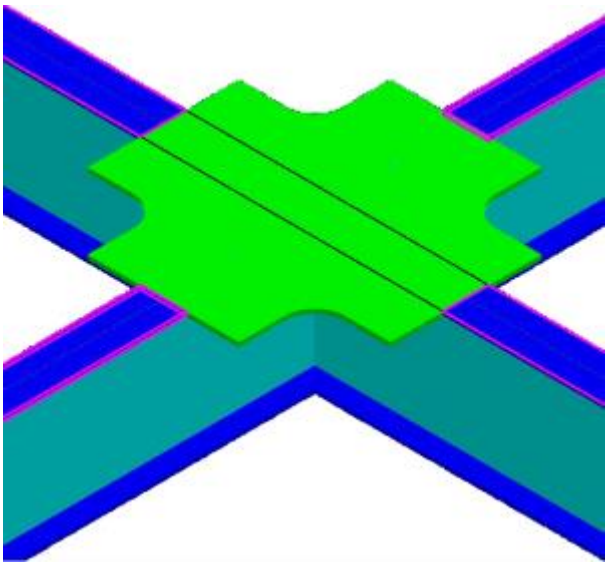
The following advanced plate system macros create insert plates, which reinforce intersecting members or supported and supporting members.

NOTE If the members share a nodal connection, use a side plate. For more information, see *Side Plates Tab (Advanced Plate Palette Dialog Box)* (on page 284).

	**Cross Insert Plate** - Creates a cross or plus-shaped insert plate between supporting and supported members. For more information, see *Cross Insert Plate* (on page 386).
	**Cross Insert Plate 1** - Creates a cross or plus-shaped insert plate between intersecting members. For more information, see *Cross Insert Plate 1* (on page 390).
	**Tee Insert Plate** - Creates a tee-shaped insert plate between supported and supporting members. For more information, see *Tee Insert Plate* (on page 394).

	**Column Insert Plate** - Creates an insert plate along a member flange and at the end of a column or brace. For more information, see *Column Insert Plate* (on page 398).
	**Column Tee Insert Plate** - Creates an insert plate between intersecting members and at the end of a column or brace. For more information, see *Column Tee Insert Plate* (on page 403).

Cross Insert Plate



Tooltip: CrossInsertPlate

Creates a cross or plus-shaped insert plate between supporting and supported members.

Cross Insert Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

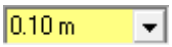
1 - Specifies a supporting member. The member can be continuous or two connected members.

2 - Specifies the first supported member.

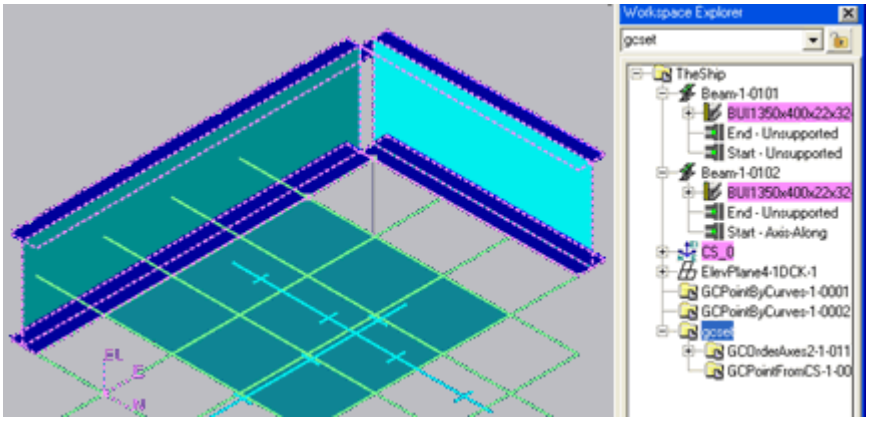
3 - Specifies the second supported member that is colinear with the first supported member.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns

to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Category - Specifies a category for the macro. **This option is not currently used.**

Length - Specifies the length of the insert plate in each direction from the intersection of the supported and supporting members.

Radius - Specifies the radius of the insert plate at each inside corner of the cross.

Direction - Specifies the flange on which to place the insert plate, relative to the member major axis. Select **Positive** to place the insert plate on the top flanges of the members. Select **Negative** to place the insert plate on the bottom flanges of the members.

What do you want to do?

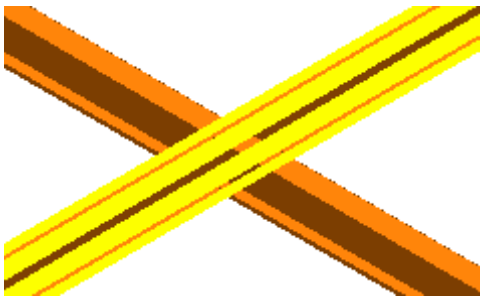
- *Place a cross insert plate on supporting and supported members* (on page 388)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a cross insert plate on supporting and supported members

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

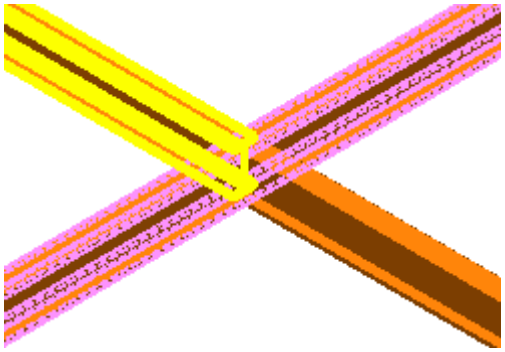
   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Insert Plate** tab, click **CrossInsertPlate**.

   *The **Cross Insert Plate** ribbon displays.*
4. In a graphic view, select the supporting member or two connected members.

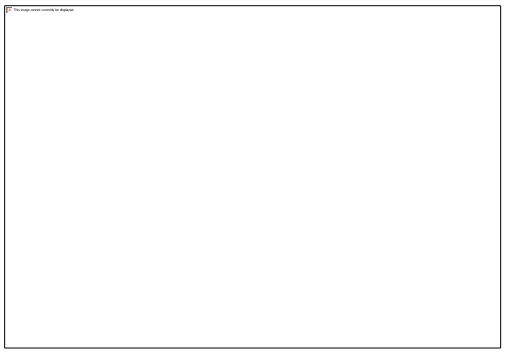


5. Click **2**.

6. Select the first supported member.

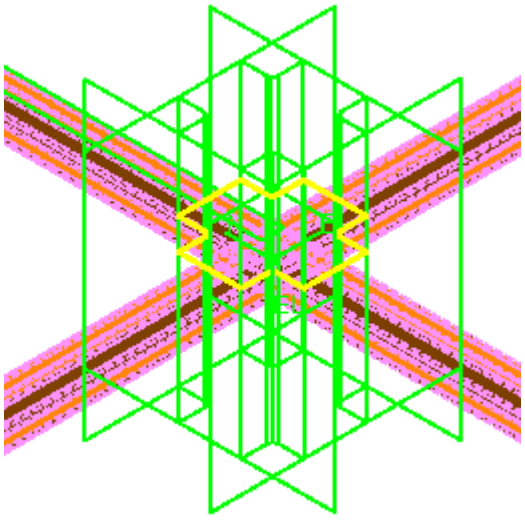


7. Select the second supported member.



8. Type the required values for **Length** and **Radius**.
9. Select the flange location of the insert plate from the **Direction** list.
10. Click **Continue**.

    *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



11. If the plate preview does not display on the correct member flange side, then:

a. Click **Select Supports** .

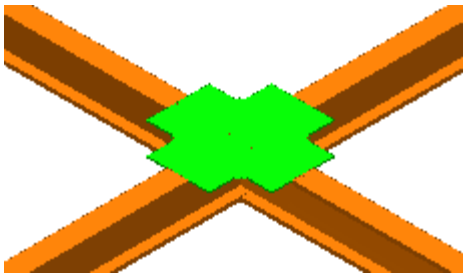
*The ribbon display returns to the **Cross Insert Plate** ribbon.*

b. Select the opposite location from the **Direction** list, click **Accept** , and then click **Continue**.

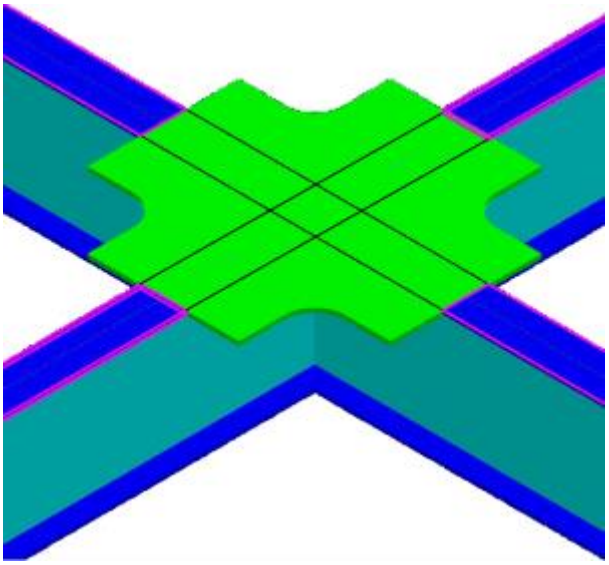
*A preview of the plate, now on the opposite member flanges, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

12. If needed, click **Define Boundaries** , and define boundaries for the plate.
13. Click **Finish**.

The software creates the insert plate.



Cross Insert Plate 1



Tooltip: CrossInsertPlate1

Creates a cross or plus-shaped insert plate between intersecting members.

Cross Insert Plate 1 Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see

Advanced Plate System Properties Dialog Box (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

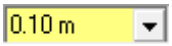
- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies the first intersecting member. The member can be continuous or two connected members.

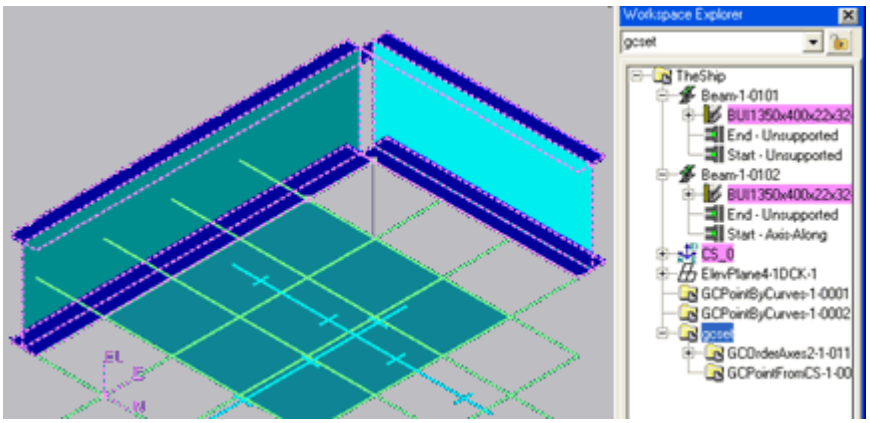
2 - Specifies the second intersecting member.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Category - Specifies a category for the macro. **This option is not currently used.**

Length - Specifies the length of the insert plate in each direction from the intersection of the intersecting members.

Width - Specifies the width of the insert plate centered along each member flange.

Radius - Specifies the radius of the insert plate at each inside corner of the cross.

Direction - Specifies the flange on which to place the insert plate, relative to the member major axis. Select **Positive** to place the insert plate on the top flanges of the members. Select **Negative** to place the insert plate on the bottom flanges of the members.

What do you want to do?

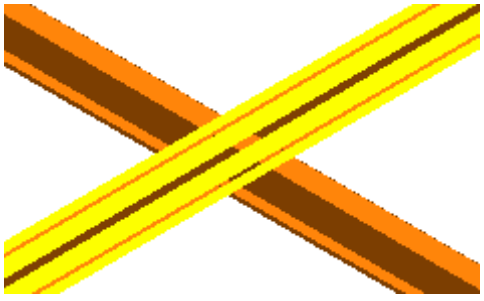
- *Place a cross insert plate on intersecting members* (on page 392)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a cross insert plate on intersecting members

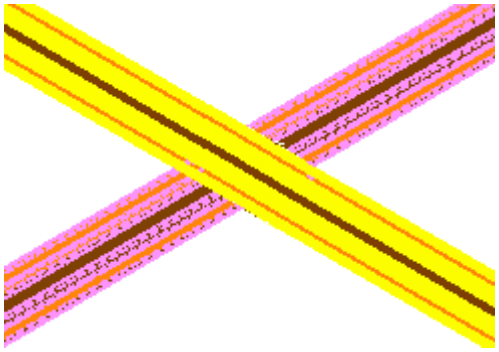
1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Insert Plate** tab, click **CrossInsertPlate1**.

   *The **Cross Insert Plate 1** ribbon displays.*
4. In a graphic view, select the first intersecting member or two connected members.

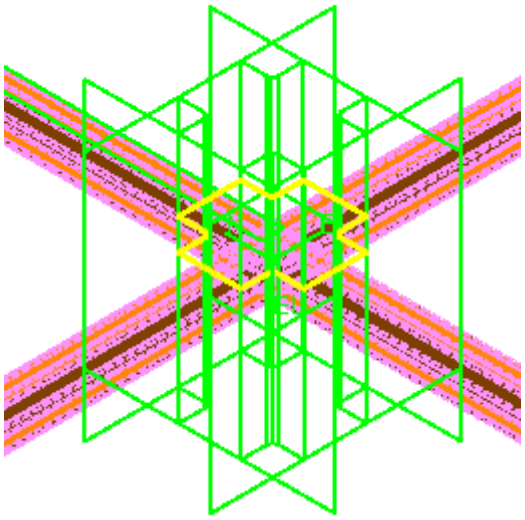


5. Click **2**.
6. Select the second intersecting member.



7. Type the required values for **Length**, **Width**, and **Radius**.
8. Select the location of the insert plate from the **Direction** list.
9. Click **Continue**.

   *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



10. If the plate preview does not display on the correct member flange side, then:
    a. Click **Select Supports** .

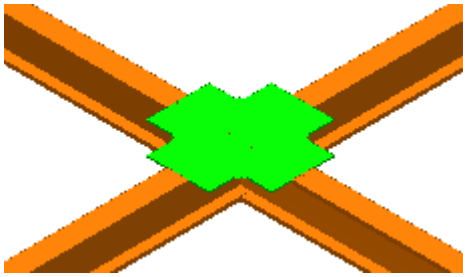
       *The ribbon display returns to the **Cross Insert Plate 1** ribbon.*

    b. Select the opposite location from the **Direction** list, click **Accept** , and then click **Continue**.

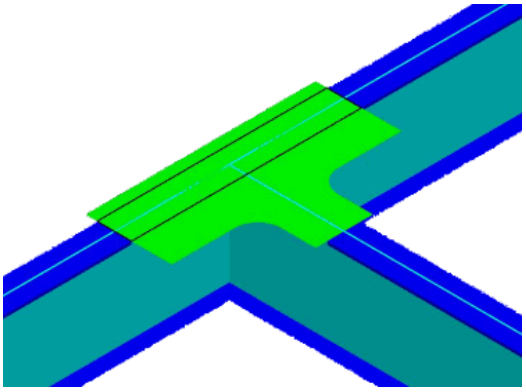
       *A preview of the plate, now on the opposite member flanges, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

11. If needed, click **Define Boundaries** , and define boundaries for the plate.
12. Click **Finish**.

The software creates the insert plate.



Tee Insert Plate



Tooltip: TInsertPlate

Creates a tee-shaped insert plate between a supported and a supporting member.

Tee Insert Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

1 - Specifies a supporting member. The member can be continuous or two connected members.

2 - Specifies the first supported member.

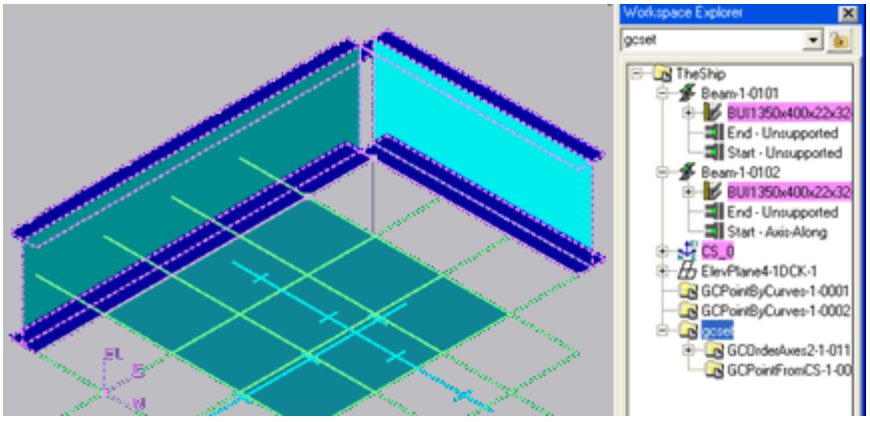
3 - Specifies the second supported member that is colinear with the first supported member.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

- 1 - No background color when no input is selected.
- 1 - Yellow background when an input is newly selected.
- 0.10 m - Yellow background when you change a parameter from the default value.
- 1 - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.

Category - Specifies a category for the macro. **This option is not currently used.**

Length - Specifies the length of the insert plate in each direction from the intersection of the supported and supporting members.

Width - Specifies the width of the insert plate centered along each member flange.

Radius - Specifies the radius of the insert plate at each inside corner of the cross.

Direction - Specifies the flange on which to place the insert plate, relative to the member major axis. Select **Positive** to place the insert plate on the top flanges of the members. Select **Negative** to place the insert plate on the bottom flanges of the members.

What do you want to do?

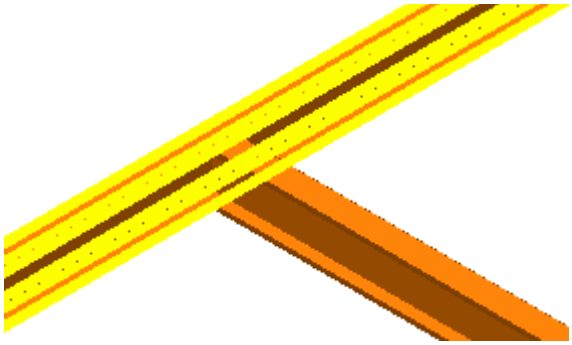
- *Place a tee insert plate on supporting and supported members* (on page 396)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a tee insert plate on supporting and supported members

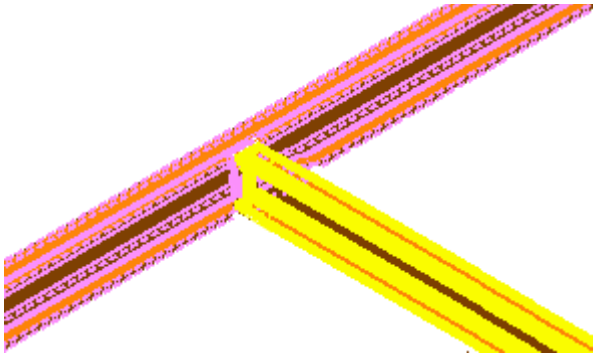
1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Insert Plate** tab, click **TInsertPlate**.

   *The **Tee Insert Plate** ribbon displays.*
4. In a graphic view, select the supporting member or two connected members.



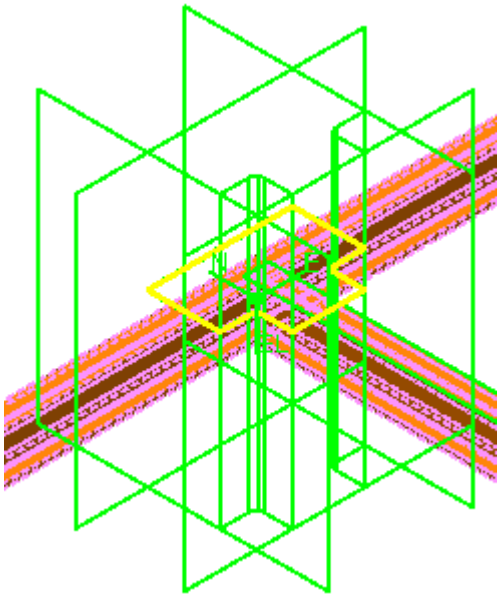
5. Click **2**.
6. Select the supported member.



7. Type the required values for **Length**, **Width**, and **Radius**.

8. Select the location of the insert plate from the **Direction** list.
9. Click **Continue**.

   *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



10. If the plate preview does not display on the correct member flange side, then:
    a. Click **Select Supports** .

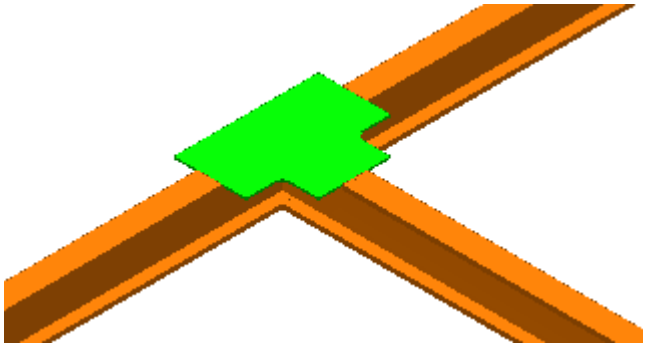
       *The ribbon display returns to the **Tee Insert Plate** ribbon.*

    b. Select the opposite location from the **Direction** list, click **Accept** , and then click **Continue**.

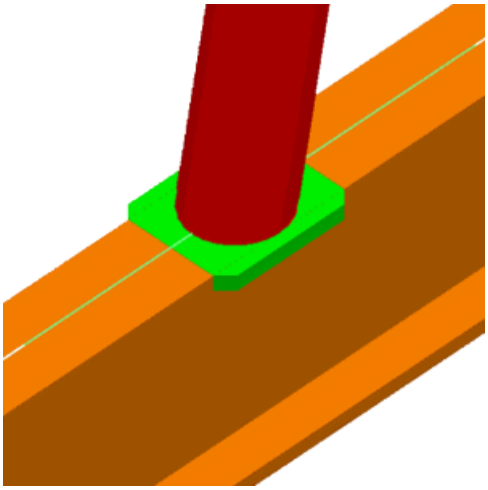
       *A preview of the plate, now on the opposite member flanges, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

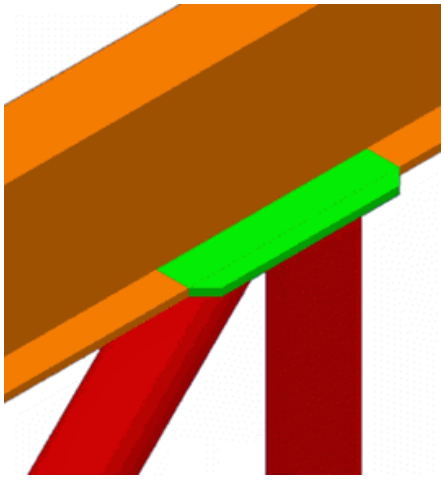
11. If needed, click **Define Boundaries** , and define boundaries for the plate.
12. Click **Finish**.

    The software creates the insert plate.



Column Insert Plate





Tooltip: InsertPlate

Creates an insert plate along a member flange and at the end of one or more columns or braces.

Column Insert Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

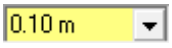
1 - Specifies the supporting beam member or edge reinforcement. The member can be a rolled member or a built-up member.

2 - Specifies the column or brace connected to the supporting member.

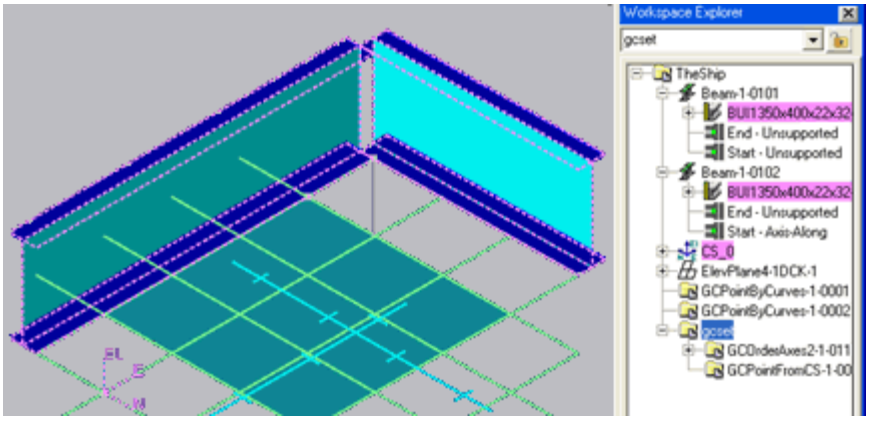
(3) - (Optional) Specifies secondary columns or braces that are also attached to the insert plate.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

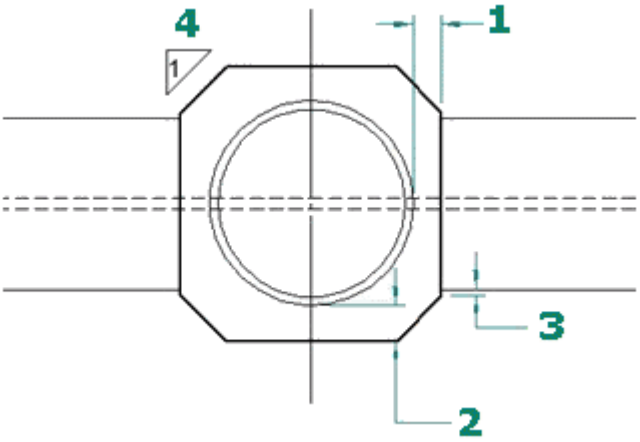
Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.



1 - Length Offset

2 - Width Offset

3 - Offset

4 - Slope

Category - Specifies a category for the macro. **This option is not currently used.**

Length Offset - Specifies the distance of the connected edge of the insert plate from the column along the length of the supporting member.

Width Offset - Specifies the distance of the free edge of the insert plate from the column.

Offset - Specifies the distance of the snipe corner from the edge of the supporting member flange.

Slope - Specifies the angle of the sniped corners.

Location - Specifies the flange on which to place the insert plate, relative to the member major axis. Select **Positive** to place the insert plate on the top flanges of the members. Select **Negative** to place the insert plate on the bottom flanges of the members.

What do you want to do?

- *Place a column insert plate along a member* (on page 400)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

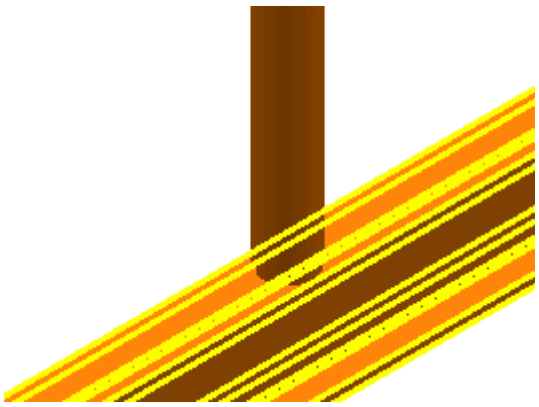
Place a column insert plate along a member

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

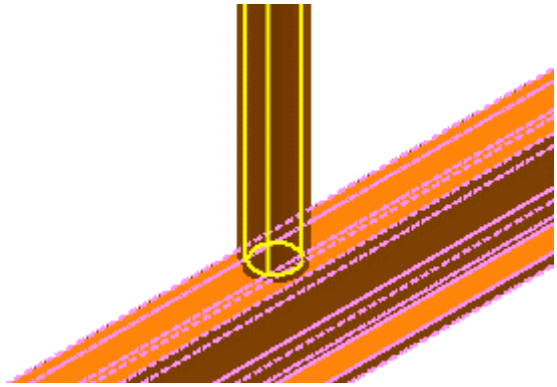
   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Insert Plate** tab, click **InsertPlate**.

   *The **Column Insert Plate** ribbon displays.*

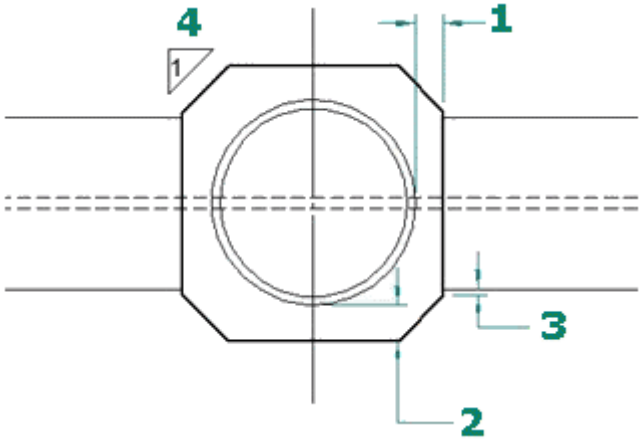
4. In a graphic view, select a continuous beam member.



5. Select the column or brace.



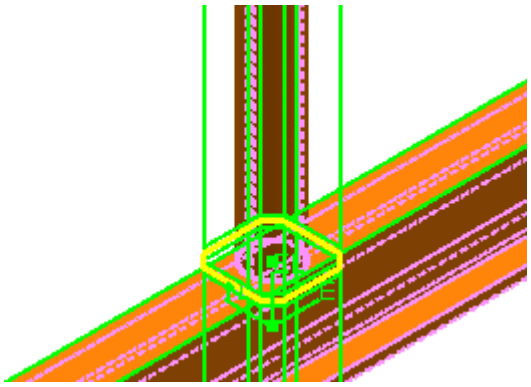
6. Optionally, click (3), and select secondary columns or braces.
7. Type the required values for **Length Offset**, **Width Offset**, **Offset**, and **Slope**.



1 - Length Offset
2 - Width Offset
3 - Offset
4 - Slope

8. Select the member side for the insert plate from the **Location** list.
9. Click **Continue**.

*A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



10. If the plate preview does not display on the correct member flange side, then:
    a. Click **Select Supports** .

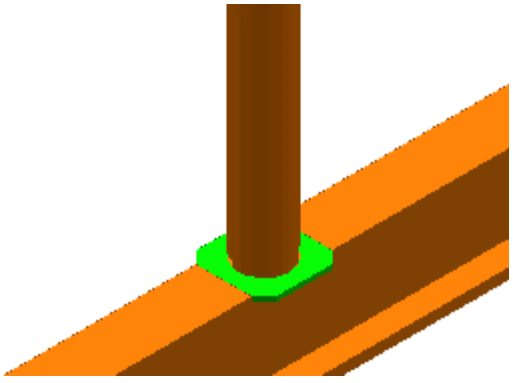
       *The ribbon display returns to the **Column Insert Plate** ribbon.*

    b. Select the opposite location from the **Location** list, click **Accept** , and then click **Continue**.

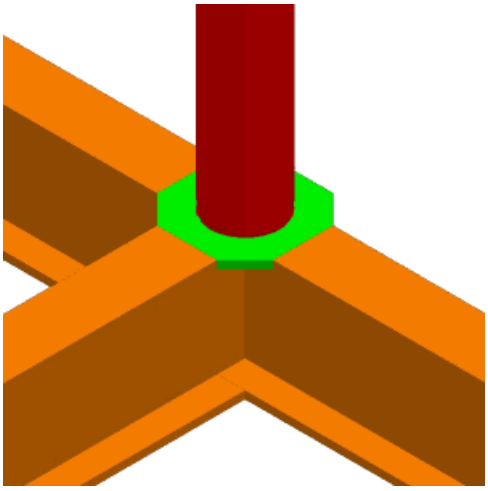
       *A preview of the plate, now on the opposite member flange, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

11. If needed, click **Define Boundaries** , and define boundaries for the plate.
12. Click **Finish**.

    The software creates the insert plate.



Column Tee Insert Plate



Tooltip: TInsertPlate1

Creates an insert plate between a supported and a supporting member and, optionally, at the end of a column or brace.

Column Tee Insert Plate Ribbon

Properties

Opens the **Advanced Plate System Properties** dialog box to view and modify the properties of the advanced plate system. The software defines the default properties from the selected structural specification and the advanced plate system rules. For more information, see *Advanced Plate System Properties Dialog Box* (on page 408).

Macro Definition

Specifies the advanced plate system macro to use. The macro that you select determines the inputs that the software requires. Select one of the following:

- Advanced plate system macros on the selected tab of the **Advanced Plate Palette** dialog box. The current macro is the default value.
- Recently selected macros.
- **Construction** - Creates a custom advanced plate system by using a basic geometric construction macro. For more information, see *Geometric Constructions* (on page 1027).
- **APS Icon Browser** - Displays the *Advanced Plate Palette Dialog Box* (on page 283).
- **More** - Displays a list of all available advanced plate system macros. For more information, see *Select Macro Dialog Box* (on page 408).

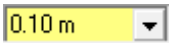
1 - Specifies the supporting beam member. The member can be a rolled member or a built-up member.

2 - Specifies the supported beam member. The member can be a rolled member or a built-up member.

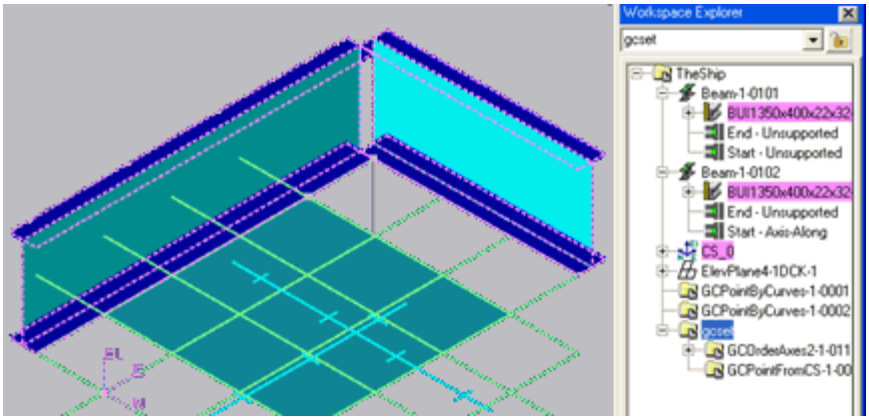
(3) - (Optional) Specifies the column or brace connected to the supporting member.

Color Coding

For a new advanced plate system, you can identify the status of input items by the background color of the step:

-  - No background color when no input is selected.
-  - Yellow background when an input is newly selected.
-  - Yellow background when you change a parameter from the default value.
-  - Blue background when you previously selected an input.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Reject

Clears the selection for the current step.

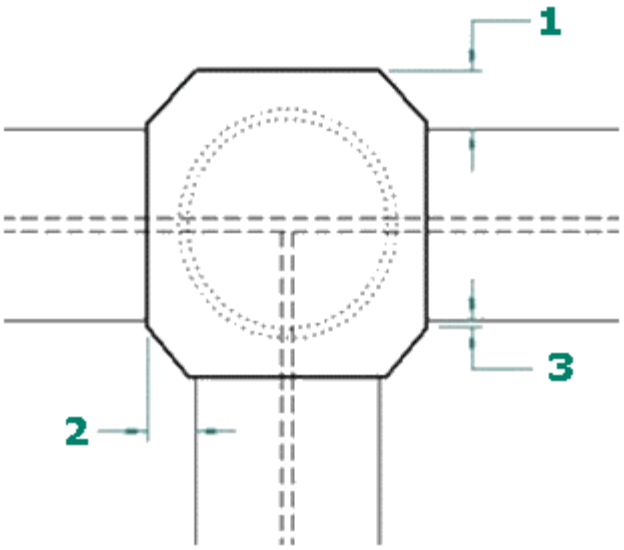
Accept

Accepts the selection for the current step.

Continue

Completes the advanced plate system definition, shows a preview of the system, and returns to the **Advanced Plate System** ribbon. For more information, see *Advanced Plate System* (on page 275).

NOTE To change the advanced plate system definition, click **Select Supports** on the **Advanced Plate System** ribbon.



1 - Offset 1

2 - Offset 2

3 - Offset

Category - Specifies a category for the macro. **This option is not currently used.**

Offset 1 - Specifies the distance of the edges of the insert plate from the flange of the supporting member.

Offset 2 - Specifies the distance of the edges of the insert plate from the flange of the supported member.

Offset - Specifies the distance of the snipe corner from the edge of the supporting and supported member flanges.

Location - Specifies the flange on which to place the insert plate, relative to the member major axis. Select **Positive** to place the insert plate on the top flanges of the members. Select **Negative** to place the insert plate on the bottom flanges of the members.

What do you want to do?

- *Place a column insert plate on supporting and supporting members* (on page 405)
- *Modify an advanced plate system* (on page 281)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Place a column insert plate on supporting and supporting members

1. Click **Advanced Plate System** on the vertical toolbar.
2. Select **APS Icon Browser** from the **Macro Definition** list.

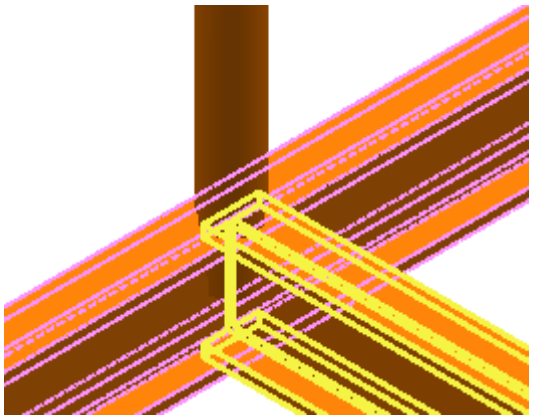
   *The **Advanced Plate Palette** dialog box displays.*
3. On the **Insert Plate** tab, click **TInsertPlate1**.

   *The **Column Tee Insert Plate** ribbon displays.*

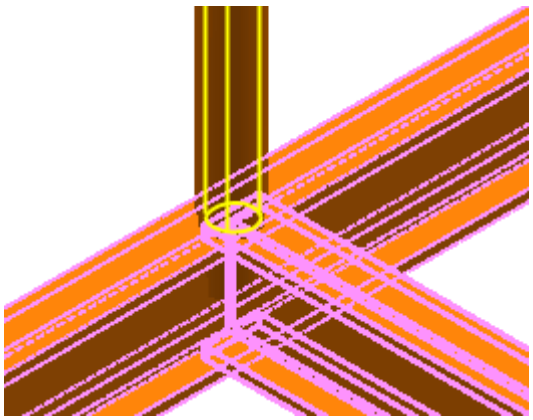
4. In a graphic view, select a continuous supporting beam member.



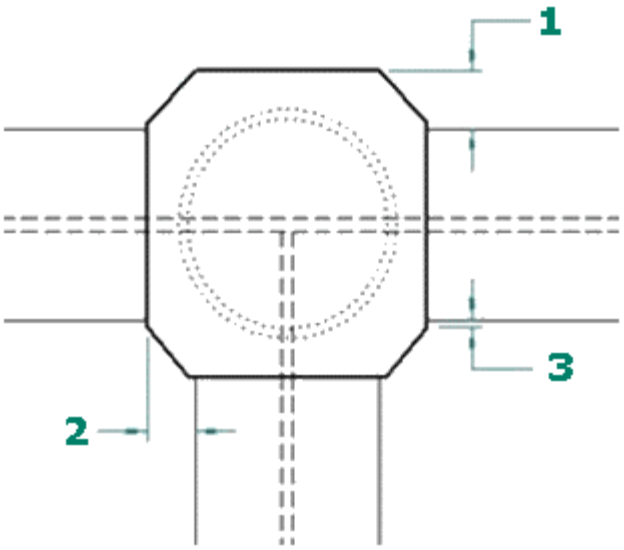
5. Select the supported beam member.



6. Optionally, click (3), and select a column or brace connected to the supporting member.



7. Type the required values for **Offset 1**, **Offset 2**, and **Offset**.



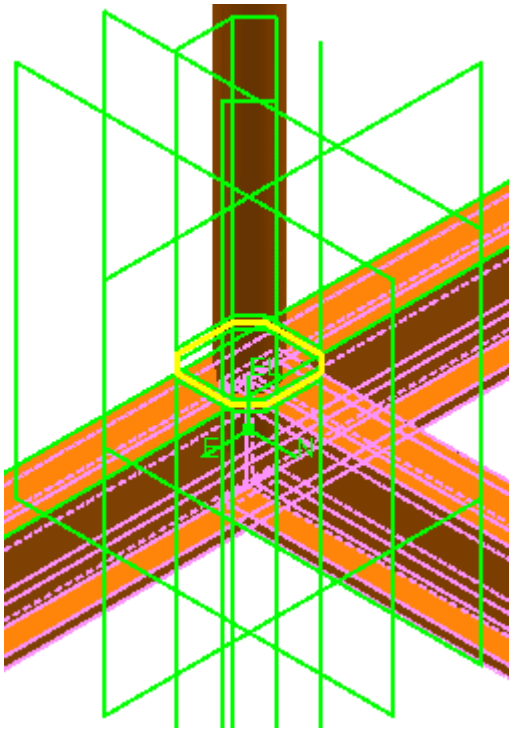
1 - Offset 1

2 - Offset 2

3 - Offset

8. Select the member side for the insert plate from the **Location** list.
9. Click **Continue**.

   *A preview of the plate and the geometric constructions used to create the plate displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*



10. If the plate preview does not display on the correct member flange side, then:
    a. Click **Select Supports** .

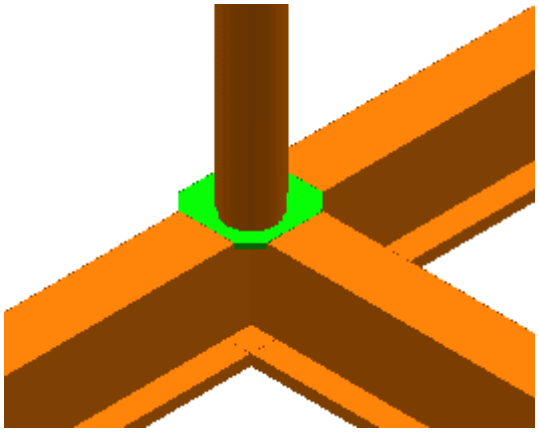
       *The ribbon display returns to the **Column Tee Insert Plate** ribbon.*

b. Select the opposite location from the **Location** list, click **Accept**, and then click **Continue**.

*A preview of the plate, now on the opposite member flange, displays in the graphic view. The ribbon display returns to the **Advanced Plate System** ribbon.*

11. If needed, click **Define Boundaries**, and define boundaries for the plate.
12. Click **Finish**.

The software creates the insert plate.



Select Macro Dialog Box

Specifies an advanced plate system macro to use. The dialog box displays when you click **More** in the **Macro Definition** list. Macros in the catalog database are displayed in a hierarchical list. After you select a macro, the appropriate options display on the **Advanced Plate System** ribbon.

This dialog box displays the same macros as the **Advanced Plate System** dialog box as a list. For information on the macro types, see *Advanced Plate Palette Dialog Box* (on page 283).

Advanced Plate System Properties Dialog Box

Specifies the properties for the root or leaf advanced plate system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.

- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Advanced Plate System Properties Dialog Box) (on page 409)

Material Tab (Advanced Plate System Properties Dialog Box) (on page 412)

Molded Conventions Tab (Advanced Plate System Properties Dialog Box) (on page 412)

Relationships Tab (Advanced Plate System Properties Dialog Box) (on page 426)

Main Tab (Advanced Plate System Properties Dialog Box)

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: <*Reference plane name*>-<*Index number*><*Plate type*>-<*Workshare location ID*>. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): <*Global CS axis*><*Global CS position along the axis in mm*>-<*Index number*><*Plate type*>-<*Workshare location ID*>, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A*-<*Unique index number*><*Plate type*>-<*Workshare location ID*>, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

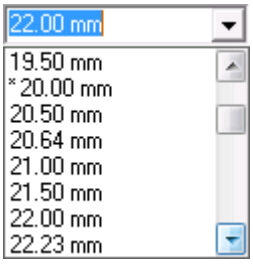
Tightness

Specifies the level of tightness as it applies to the entire plate system.

Active - Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Advanced Plate System Properties Dialog Box)

Specifies the default material settings for the advanced plate system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

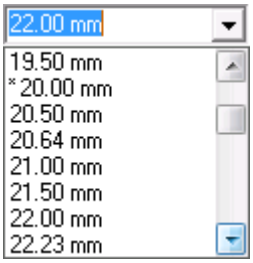
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Advanced Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the advanced plate system.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



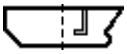
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



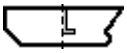
Aft - Orients profile flanges aft.



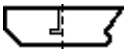
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



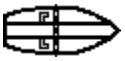
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



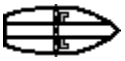
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



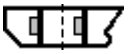
Outboard - Positions thickness away from the forward/aft reference.



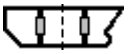
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

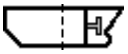
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



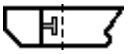
Aft - Orients profiles aft.



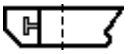
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



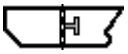
Aft - Orients profiles aft.



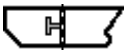
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



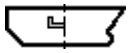
Starboard - Orients profile flanges to the starboard side.



Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



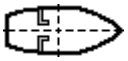
Aft - Orients profile flanges aft.



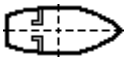
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



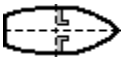
Aft - Orients profile flanges aft.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the

plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for

designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

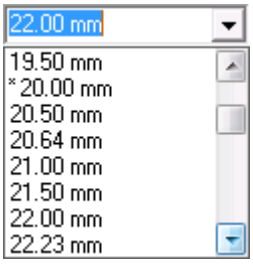


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Relationships Tab (Advanced Plate System Properties Dialog Box)

Displays a list of objects related to the plate system. Select **General**, **Boundaries**, **Bounds**, or **Connections** as the relationship category from the list on the right.

List color - Specifies the display color in the graphic view for all items in the list.

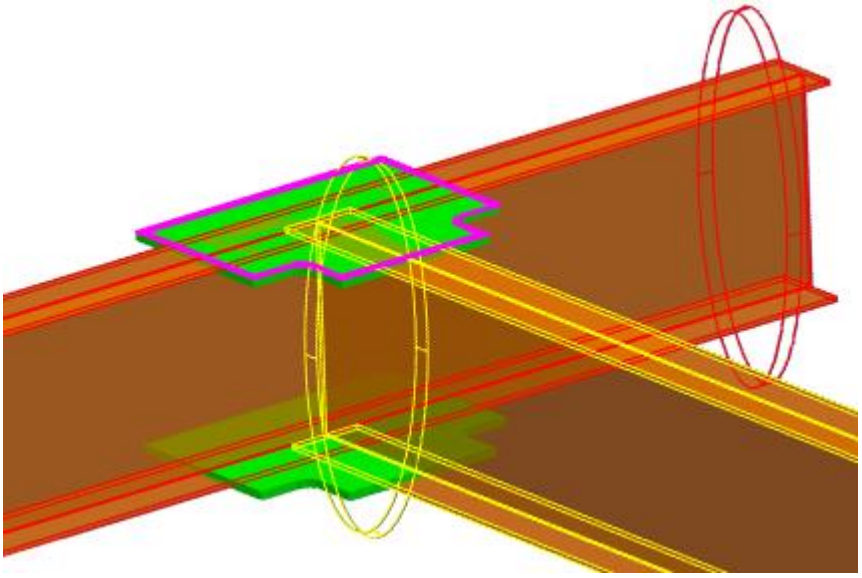
Item color - Specifies the display color in the graphic view for a selected item in the list.

Name - Displays the name of the related object.

Type - Displays the type of related object.

Go To - Displays the properties of the selected object.

In the following example, the advanced plate system in magenta, the list color is red, and the color of the item selected in the list is yellow.



Child Plate System

The **Child Plate System** command creates a plate system that is a child to and in the same plane as another plate system. As a child plate system, it inherits the parent plate system's category, reference plane, molded conventions, material properties, tightness, topological boundaries, and continuity.

Child Plate System Ribbon

Displays the options that you use to create a child plate system in the model.

Plate System Properties

Opens the **Child Plate System Properties** dialog box, which you use to view and modify the properties of the plate system. The initial plate system properties default from the parent

plate system, if there is one, or from the selected structural specification. For more information, see *Child Plate System Properties Dialog Box* (on page 432).

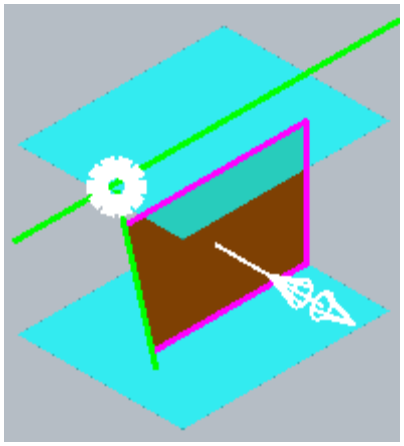
Select Surface

Specifies an existing plate system in the model that you want to make a child plate system.

Define Boundaries

Defines the outside boundaries of the child plate system. You can define the boundaries by selecting objects in the model or by drawing the boundaries. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See the *Define Boundary Controls* and *Define Plane Controls* sections below.

NOTE When selecting an existing plate system, if displays for **Define Boundaries**, then the boundaries do not form a closed shape because a boundary has been modified. This condition most commonly occurs when the plate system has a sketched boundary that no longer intersects a plate system or reference plane boundary. Click **Define Boundaries** to display the location of the gap in the boundary.



Finish

Places the system using the defined parameters.

Define Boundary Controls

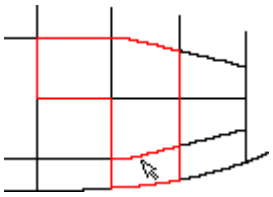
Available when you use the **Define Boundaries** option.

Pick Boundaries

Specifies objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to clarify the needed boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Select this option to draw the boundaries. This option is only available when you are defining the boundaries.

Sketching Plane

Select the sketching plane for the boundary.

Add Intersecting Item

Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.

Add Projection Item

Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the boundary.

Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Project Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

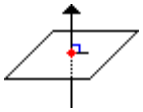
Define Plane Controls

Available when you use the **Boundary Curve** option.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only

available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

What do you want to do?

- *Create a child plate system* (on page 431)
- *Modify a plate system* (on page 104)
- *Delete a plate system* (on page 106)

Create a child plate system

1. Click **Child Plate System** command.
2. Select the parent plate system.
3. Click **Define Boundaries**.
4. Define the child plate system boundaries using one or more of the following methods:

   Pick boundaries (on page 81)
   Use the Boundary List dialog box (on page 81)
   Define a boundary curve (on page 82)
5. Click **Accept**.
6. If the boundary definition results in more than one solution, you must select bounded areas that together compose the total bounded area. Pick the needed solutions using the following procedure

   Solve ambiguous solution created by selected boundaries (on page 82)
7. Set the remaining child plate system properties.
8. Click **Finish**.

NOTES

- The new child plate system references, but does not copy, the surface geometry of the selected parent plate system.
- If you change the parent surface geometry, all child plate systems that reference that surface geometry also change.

Child Plate System Properties Dialog Box

Specifies the properties for the root or leaf child plate system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Child Plate System Properties Dialog Box) (on page 432)

Material Tab (Child Plate System Properties Dialog Box) (on page 435)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Weight & CG Tab (on page 129)

Extended User Attributes Tab (on page 129)

Routing Tab (on page 129)

Main Tab (Child Plate System Properties Dialog Box)

Specifies the general properties of the child plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

Specifies the level of tightness as it applies to the entire plate system.

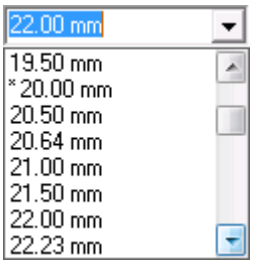
Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid

connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Child Plate System Properties Dialog Box)

Specifies the material properties for the child plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

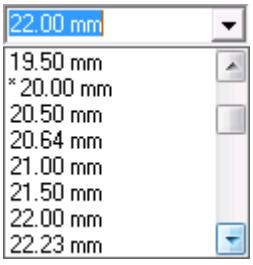
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Child Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the child plate system that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



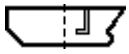
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



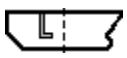
Aft - Orients profile flanges aft.



Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



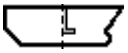
Aft - Orients profile flanges aft.



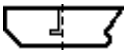
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



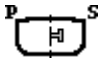
Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



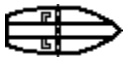
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



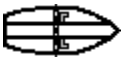
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



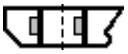
Outboard - Positions thickness away from the forward/aft reference.



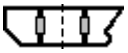
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

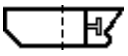
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



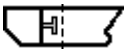
Aft - Orients profiles aft.



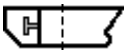
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



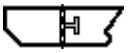
Aft - Orients profiles aft.



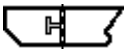
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



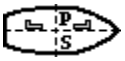
Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

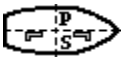
Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



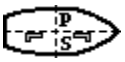
Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



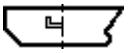
Starboard - Orients profile flanges to the starboard side.



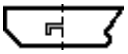
Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



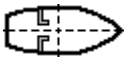
Aft - Orients profile flanges aft.



Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



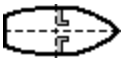
Aft - Orients profile flanges aft.



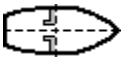
Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

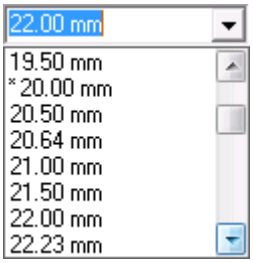


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Ruled Plate System

The **Ruled Plate System** command creates a plate system that is interpolated between two non-planar curves, providing a smooth transition. You have the option of selecting existing curves in the model or drawing the curves in the Sketch 2D environment. A common use for this command is to place a plate system between two other plate systems that form a corner.

Ruled Plate System Ribbon

Displays the options that you use to create a ruled plate system in the model.

Ruled Plate System Properties

Opens the **Ruled Plate System Properties** dialog box, which you use to view and modify the properties of the plate system that you are about to place in the model. The initial plate system properties default from the parent plate system, if there is one, or from the selected structural specification. For more information, see *Ruled Plate System Properties Dialog Box* (on page 456).

First Curve Definition

Defines the first curve of the ruled plate system. See the *Define Curve Controls* and *Define Plane Controls* sections below.

Second Curve Definition

Defines the second curve of the ruled plate system. See the *Define Curve Controls* and *Define Plane Controls* sections below.

Define Boundaries

Define the outside boundaries of the plate system. You can define the boundaries by selecting objects in the model or by drawing the boundaries. If you select objects in the model and those objects are moved, the software automatically resizes the plate system to maintain the boundary relationship. See the *Define Boundary Controls* and *Define Plane Controls* sections below.

Finish

Places the system using the defined parameters.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Parent System

Specifies the parent system for the plate system that you are placing. If the needed system is not available, click **More** to open the **Select System** dialog box. For more information, see *Select System Dialog Box* in the *Common User's Guide*.

NOTE Valid systems are created in the Systems and Specifications task or with the **New System** command. For more information see the *Systems and Specifications User's Guide,* or *New System* in the *Common User's Guide*.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part

that is a child to the profile system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied. Values are specific to the **Type** value. For more information, see the molded conventions for each plate type in *Molded Conventions Tab (Planar Plate System Properties Dialog Box)* (on page 110).

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Define Curve Controls

Available when you use the **First Curve Definition** and **Second Curve Definition** options.

 **Pick Curve**

Select this option to select an existing curve in the model as a curve for a ruled plate system.

 **Sketch Curve**

Select this option to draw a curve for the ruled plate system.

 **Sketching Plane**

Specify the sketching plane for the curve. This is the first step in defining the curve. See the *Define Plane Controls* section below.

 **Add Intersecting Item**

Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.

 **Add Projection Item**

Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.

 **Sketch 2D**

Opens the Sketch 2D environment in which you can draw the curve.

NOTE To display a grid or reference plane ruler in the Sketch 2D environment, first define an active coordinate system using **Tools** > **PinPoint**. For more information, see PinPoint in the *Common User's Guide* and *2D Grids Ruler* in the *SmartSketch Drawing Editor Help*.

Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Project Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Define Boundary Controls

Available when you use the **Boundary Curve** option.

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

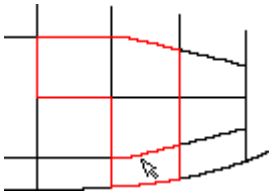
TIP Pressing the SHIFT key selects the **Pick Edges** option after you have selected a boundary plate. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Select this option to pick free edges on plates to define the boundary. This option is available if you selected a plate in the **Pick Boundaries** step.

Solve Ambiguity

If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to clarify the needed boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Select this option to draw the boundaries. This option is only available when you are defining the boundaries.

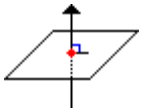
Define Plane Controls

Available when you use the **First Curve Definition**, **Second Curve Definition** and **Boundary Curve** options.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only

available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

What do you want to do?

- *Create a ruled plate system using existing curves* (on page 454)
- *Create a ruled plate system by drawing curves* (on page 455)
- *Modify a plate system* (on page 104)
- *Move a plate system to a different system in the model hierarchy* (on page 105)
- *Delete a plate system* (on page 106)

Create a ruled plate system using existing curves

1. Click **Ruled Plate System** .
2. Click **First Curve Definition** .

   Pick Curve *is selected as the default option.*
3. Select the first curve.
4. Click **Second Curve Definition** .

   Pick Curve *is selected as the default option.*
5. Select the second curve.
6. Click **Define Boundaries** .
7. Define the ruled plate system boundaries using one or more of the following methods:

   Pick boundaries (on page 81)
   Define a boundary curve (on page 82)
8. Click **Accept** .
9. If the boundary definition results in more than one solution, you must select bounded areas that together compose the total bounded area. Pick the needed solutions using the following procedure:

Solve ambiguous solution created by selected boundaries (on page 82)

10. Set the remaining plate system properties.
11. Click **Finish**.

Create a ruled plate system by drawing curves

1. Click **Ruled Plate System** .
2. Click **First Curve Definition** .
3. Click **Sketch Curve** .
4. Specify the sketching plane for the curve using one of the following methods:

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)

5. Click **Add Intersecting Item** .
6. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.
7. Click **Add Projection Item** .
8. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.
9. Click **Sketch 2D** .
10. In the Sketch 2D environment, sketch the first curve for the ruled plate system.
11. Click **Finish** in the Sketch 2D environment.
12. Click **Second Curve Definition** .
13. Click **Sketch Curve** .
14. Specify the sketching plane for the second curve using one of the following methods:

    Define a coincident plane (on page 76)
    Define an offset plane (on page 77)
    Define a plane using a point and a normal vector (on page 78)
    Define a plane using three points (on page 79)

15. Click **Add Intersecting Item** .
16. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.
17. Click **Add Projection Item** .
18. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.
19. Click **Sketch 2D** .

20. In the Sketch 2D environment, sketch the second curve for the ruled plate system.
21. Click **Finish** in the Sketch 2D environment.
22. Click **Define Boundaries** .
23. Define the ruled plate system boundaries using one or more of the following methods:

    Pick boundaries (on page 81)
    Define a boundary curve (on page 82)
24. Click **Accept** .
25. If the boundary definition results in more than one solution, you must select bounded areas that together compose the total bounded area. Pick the needed solutions using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
26. Set the remaining plate system properties.
27. Click **Finish**.

Ruled Plate System Properties Dialog Box

Specifies the properties for the root or leaf plate system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Ruled Plate System Properties Dialog Box) (on page 457)

Material Tab (Ruled Plate System Properties Dialog Box) (on page 459)

Molded Conventions Tab (Ruled Plate System Properties Dialog Box) (on page 460)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Main Tab (Ruled Plate System Properties Dialog Box)

Specifies the general properties of the ruled plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: <*Reference plane name*>-<*Index number*><*Plate type*>-<*Workshare location ID*>. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): <*Global CS axis*><*Global CS position along the axis in mm*>-<*Index number*><*Plate type*>-<*Workshare location ID*>, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A*-<*Unique index number*><*Plate type*>-<*Workshare location ID*>, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane:* <*Global CS axis*> = <*Global CS position along the axis*>, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: <*Parent system name*>-<*Object type*>. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the

plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

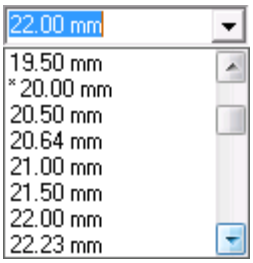
Specifies the level of tightness as it applies to the entire plate system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Ruled Plate System Properties Dialog Box)

Specifies the material properties for the ruled plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

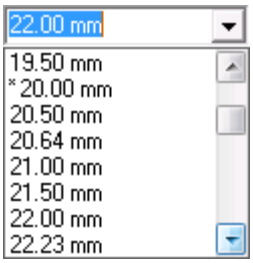
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Ruled Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the ruled plate system that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



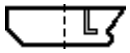
Starboard - Orients profile flanges to the starboard side.



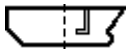
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



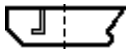
Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



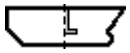
Aft - Orients profile flanges aft.



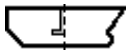
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



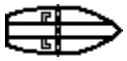
Aft - Orients profile flanges aft. This is the default value.



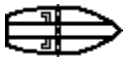
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

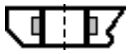
Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.



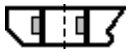
Outboard - Positions thickness away from the forward/aft reference.



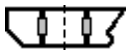
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

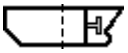
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



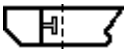
Aft - Orients profiles aft.



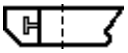
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



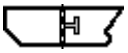
Aft - Orients profiles aft.



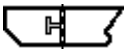
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



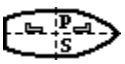
Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

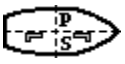
Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



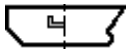
Starboard - Orients profile flanges to the starboard side.



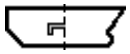
Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

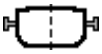
Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



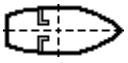
Aft - Orients profile flanges aft.



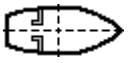
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



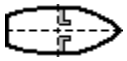
Aft - Orients profile flanges aft.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube

member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

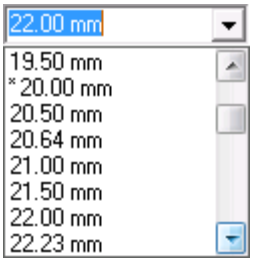


Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

SECTION 4

Openings

 Places an opening in a plate system created in the Molded Forms task. An opening is a cutout in a plate system that is part of the high-level design. You can specify the shape of the opening by:

- Sketching it in the Sketch 2D environment. This method provides the most flexibility for creating a small number of openings or for sketching unusual opening shapes. The openings can be dimensionally constrained to other objects in the model.
- Selecting a predefined advanced opening macro from the catalog and placing it directly in the 3D model. The opening is defined by a set of supports. This methods provides the most speed when placing large numbers of openings that share similar constraints.

Opening Properties

When you place an opening, you can define opening properties to better reflect the more granular details of the purpose of the opening or hole, when it is cut, and how it is produced. These properties are grouped into the following categories:

Design Properties

Represent the functional purpose of the opening or hole. The design property reflects why the opening exists in the model. Typical design properties include:

- Access (Permanent)
- Access (Temporary)
- Lightening (Weight Reduction)
- Air Escape
- Liquid Escape
- Air/Liquid Escape
- Foothold
- Martyr

Planning Properties

Represent the staging of the opening or hole. The planning property denotes the point in the design and production lifecycle when the cut is physically made. Typical planning properties include:

- Design
- Shop (Assembly)
- Ship (Field)

Production Properties

Represent the production method of the opening or hole. The planning property denotes whether the cut is truly cut from the system or part, whether it is marked, bridged, and so

forth. Typical production properties include:

- Cut (Default)
- Mark
- Bridge
- Chill Hole (Patch)

NOTES

- You can place multiple openings at one time on a plate system or copy openings from one plate system to another. For more information, see *Model Bulkheads with Multiple Openings* (on page 57).
- Minimum height or distances are maintained by the software for objects that use Sketch 2D, such as sketched openings. For more information, see *Minimum Distance* (on page 92).
- When you create or edit an opening, the dialog box title displays the plate system name using the format **[Plate System*: <name of the plate system>*]**.



- Openings created in the Molded Forms task appear in conceptual drawings such as classification scantlings, whereas cutouts and holes created in the Hole Management and Structural Detailing tasks do not appear in the conceptual drawings. The difference between an opening and a hole is an important distinction primarily for the Drawings and Reports task.

Openings Ribbon

Specifies the properties for the opening that you are placing.

Opening Properties

Opens the **Opening Properties** dialog box, which is used to define properties of the opening that are not available on the ribbon. For more information, see *Opening Properties Dialog Box* (on page 482).

Plate System

Selects the plate system on which to place the opening.

Define using 2D sketcher

Defines the opening by sketching in the Sketch 2D environment. See **2D Sketcher Options**, below.

NOTE To display a grid or reference plane ruler in the Sketch 2D environment, first define an active coordinate system using **Tools** > **PinPoint**. For more information, see PinPoint in the *Common User's Guide* and *2D Grids Ruler* in the *SmartSketch Drawing Editor Help*.

Define by catalog selection

Defines an advanced opening directly in the 3D model by selecting an opening geometric construction and defining its dimensions and supports. This option opens the **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box.

Opening Palette Dialog Box (on page 482)

Geometric Construction Explorer (on page 703)

Finish

Places the opening using the defined parameters.

2D Sketcher Options

Sketching Plane

Specify the sketching plane for the opening. This is the first step in defining the opening.

Add Intersecting Item

Allows you to select objects that intersect the sketching plane in the 3D environment. You see the selected objects in the Sketch 2D environment.

Add Projection Item

Allows you to select objects that do not intersect the sketching plane in the 3D environment. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the opening.

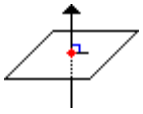
Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the

distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Reject

Clears all selections.

Accept

Accepts all selections.

Opening design type

Specifies the design opening property for the opening or hole. The default value is **Permanent Access**. The list is defined by the CutoutDesignType codelist.

Opening planning method

Specifies the planning opening property for the opening or hole. The default value is **Design**. The list is defined by the CutoutPlanningMethod codelist.

Opening production method

Specifies the production opening property for the opening or hole. The default value is **Cut**. The list is defined by the CutoutProductionMethod codelist.

NOTE For more information about opening properties, see *Openings* (on page 475).

Opening Palette Dialog Box **(on page 482)**

What do you want to do?

- *Place an opening by sketching* (on page 479)
- *Place an advanced opening* (on page 480)
- *Delete advanced opening geometry* (on page 489)
- *Modify an opening* (on page 480)
- *Copy and paste a sketched opening* (on page 481)
- *Copy and paste an advanced opening* (on page 490)
- *Delete an opening* (on page 481)

Place an opening by sketching

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define using 2D sketcher**.
4. Specify the sketching plane for the opening using one of the following methods:

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)

   TIP If you selected a planar plate system, its surface is automatically specified as the coincident sketch plane. You can specify another sketch plane by one of the other methods if needed.
5. Click **Add Intersecting Item**.
6. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in Sketch 2D as reference geometry.
7. Click **Add Projection Item**.

8. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in Sketch 2D as reference geometry.
9. Click **Sketch 2D** .

   The Sketch 2D environment opens.
10. Using the available drawing commands in the 2D environment, draw the opening.

    TIPS

    - You can draw more than one opening. Each opening is created as a separate object, but the openings share the same Sketch 2D sketch.
    - You can constrain the opening to geometric construction points offset from plate systems. For more information, see *Point at Minimum Distance Command* in the *SmartSketch Drawing Editor Drawing Editor Help*, available within the 2D environment.

    IMPORTANT You must sketch a closed contour. See *Sketch 2D Best Practices* (on page 65) for more information.
11. Use the commands on the **Relationship** and **Dimension** toolbars to constrain the opening to reference geometry, if needed.
12. Click **Finish** on the ribbon bar in the 2D environment.

    The software returns to the 3D environment.
13. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
14. Click **Finish**.

    The opening is created and appears in the graphic view and the **Workspace Explorer**.

Place an advanced opening

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection** .

   The ***Opening Palette*** *dialog box and the* ***Geometric Construction Explorer*** *dialog box display.*
4. In the **Opening Palette** dialog box, select a tab for the needed opening type, and then select an opening macro.
5. Follow the help for the selected macro. For more information, see *Opening Palette Dialog Box* (on page 482).

Modify an Opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.

3. Select the opening to modify. You can select the opening in a graphic view or from the **Workspace Explorer**.
4. Using the ribbon controls, edit the opening as needed.

TIP If multiple sketched openings were created on the same sketch, you can modify them at the same time.

Copy and paste a sketched opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the opening to copy. You can select the opening in a graphic view or from the **Workspace Explorer**.
4. Click **Edit** > **Copy**.
5. Click **Edit** > **Paste**.
6. Select **Parent of** in the **Paste** dialog box.
7. Select the plate system to receive the opening copy. You can select the plate system in a graphic view or from the **Workspace Explorer**.
8. If you want to select a new sketching plane, select **StructSketchInput** in the **Paste** dialog box.
9. Select a planar plate system, planar surface of an extruded plate system, or grid plane to use as a sketching plane. You can select a plate system or gird plane in a graphic view or from the **Workspace Explorer**. You must select the planar surface of an extruded plate system from a graphic view.
10. Click **OK**.

IMPORTANT

- **Parent for Naming** and **StructSketchInput** must be parallel to the parent plate system of the original opening.
- The original opening geometry is pasted to the new sketching plane in the same planar coordinates used on the original sketching plane.
- Only the original opening geometry is copied. Dimension and relationship constraints on the original sketch are ignored. Use **Copy Similar** to copy constraints. For more information, see *Copy by Family* (on page 1255).
- The copied opening has no relationship to the original opening. Each opening can be modified independently. Use **Copy Similar** if you need a relationship between the openings. For more information, see *Copy by Family* (on page 1255).

Delete an opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the opening to delete.
4. Click **Delete** .

Opening Properties Dialog Box

Specifies the properties for the opening.

Main Tab (Opening Properties Dialog Box) (on page 482)

Relationship Tab (on page 123)

Legacy Catalog Tab (Opening Properties Dialog Box) (on page 482)

Main Tab (Opening Properties Dialog Box)

Name - Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule - Select the naming rule to use to name the opening.

- Select **StdHierarchyChildNamingRule** to use the following syntax: <Parent Name>-<"Opening Entity"><Index Number>. For example, D0-1DCK-1-Opening Entity1, where D0 is the reference plane, 1DCK is an index number appended by the plate type (Deck in this example), 1 is the Workshare Location ID, and Opening Entity1 is Opening Entity appended by an index number.
- Select **User Defined** if you want to name the opening yourself using the appropriate box.

Opening design type

Specifies the design opening property for the opening or hole. The default value is **Permanent Access**. The list is defined by the CutoutDesignType codelist.

Opening planning method

Specifies the planning opening property for the opening or hole. The default value is **Design**. The list is defined by the CutoutPlanningMethod codelist.

Opening production method

Specifies the production opening property for the opening or hole. The default value is **Cut**. The list is defined by the CutoutProductionMethod codelist.

NOTE For more information about opening properties, see *Openings* (on page 475).

Legacy Catalog Tab (Opening Properties Dialog Box)

This tab is provided *only* for compatibility with catalog openings created in a model migrated from earlier versions of the software.

To create new catalog openings, use **Define by catalog selection**  . For more information, see *Openings* (on page 475).

Opening Palette Dialog Box

Specifies an advanced opening. The software creates an advanced opening by using a geometric construction macro. The **Opening Palette** dialog box displays when you click **Define by catalog selection**  on the **Openings** ribbon. Each tab of the palette graphically displays

options available for a type of opening macro in the Catalog database. After you select a macro, the appropriate options display on the **Advanced Openings** ribbon.

Filters

Limits the available macros on a tab to the selected type.

Clear Recent

Removes recently selected macros from the **Recent** tab.

Help

Displays help for the selected macro in a preview window. If no help is available, a preview of the selected macro displays.

NOTE You can also select **Tools** > **Automatic Preview** to display the preview window.

Custom Shape Tab (Opening Palette Dialog Box) (on page 484)

Support and Offset Tab (Opening Palette Dialog Box) (on page 505)

0S Circle Tab (Opening Palette Dialog Box) (on page 515)

3S Circle Tab (Opening Palette Dialog Box) (on page 529)

4S Circle Tab (Opening Palette Dialog Box) (on page 555)

0S Oval Tab (Opening Palette Dialog Box) (on page 576)

3S Oval Tab (Opening Palette Dialog Box) (on page 589)

4S Oval Tab (Opening Palette Dialog Box) (on page 615)

0S Rectangle Tab (Opening Palette Dialog Box) (on page 637)

3S Rectangle Tab (Opening Palette Dialog Box) (on page 653)

4S Rectangle Tab (Opening Palette Dialog Box) (on page 680)

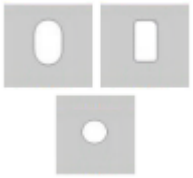
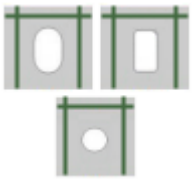
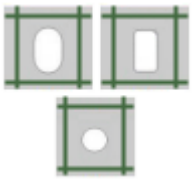
NOTE The **Recent** tab displays macros recently selected on the other tabs. Click **Clear Recent** to remove interfaces from this tab.

The **Geometric Construction Explorer** also displays. As openings are created, a list of opening macros and output opening geometry display.

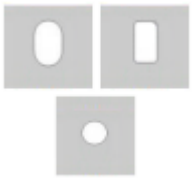
Geometric Construction Explorer **(on page 703)**

Custom Shape Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to select the shape and size of the opening based on the number of supports.

	**Advanced Opening with No Supports** - Creates an opening with no supports that you position with X, Y, and Z coordinates. The default position is the center of the plate. You can select the opening shape (circle, oval, or rounded rectangle). For more information, see *Advanced Opening with No Supports* (on page 484).
	**Advanced 3S Opening** - Creates an opening with three supports. The opening is positioned by a dimension from the first support and centered between the second and third support. You can select the opening shape (circle, oval, or rounded rectangle). For more information, see *Advanced 3S Opening* (on page 490).
	**Advanced 4S Opening** - Creates an opening with four supports and centered between the supports. You can select the opening shape (circle, oval, or rounded rectangle). For more information, see *Advanced 4S Opening* (on page 498).

Advanced Opening with No Supports



Tooltip: AOWithNoSupports

Creates an opening with no supports and positioned by X, Y, and Z coordinates. The default position is the center of the plate. You can select the opening shape (circle, oval, or rounded rectangle), and then define the parameters for that shape.

Advanced Opening with No Supports Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

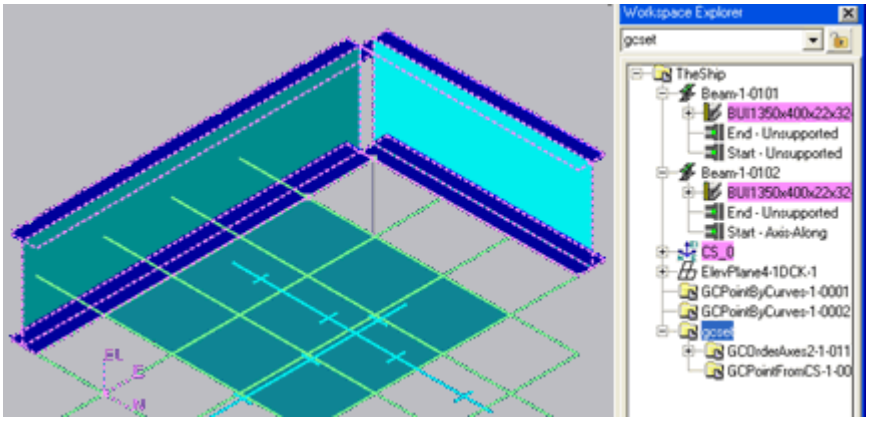
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

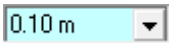
- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.

-  and  - Blue background when the software changes a value to a new suggested value.

 **Reject** - Clears the selections for the current step.

 **Accept** - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. Select **Circle**, **Oval**, or **Rounded Rect**.

Radius - Defines the radius of a circular opening.

Length - Defines the length of an oval or rounded rectangle opening.

Width - Defines the width of an oval or rounded rectangle opening.

Rounded Rad - Defines the corner radius of a rounded rectangle opening.

What do you want to do?

- *Place an advanced opening with no supports* (on page 487)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

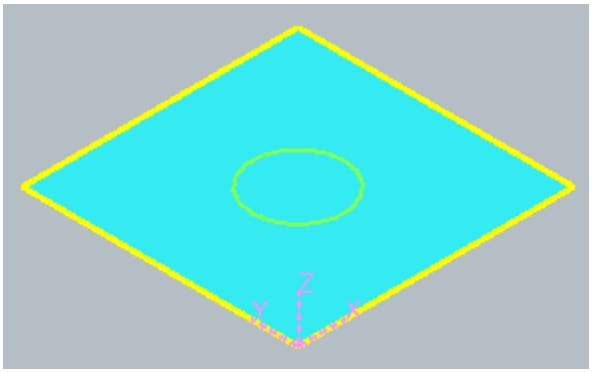
Place an advanced opening with no supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **Custom Shape** tab, click **AOWithNoSupports**.

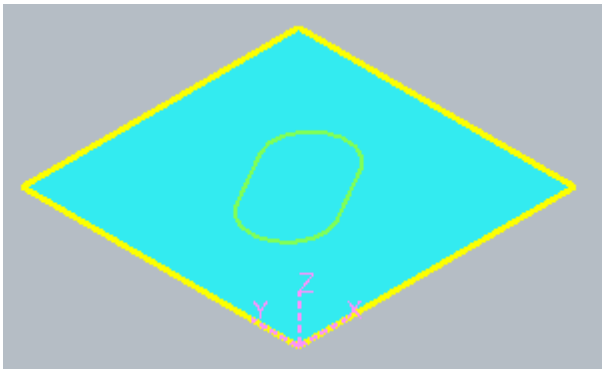
   The ribbon for the macro displays.
5. Select a coordinate system.

   *A local coordinate system at the plate origin displays. A preview of the opening using the default shape and size values displays at the center of the plate. **X**, **Y**, and **Z** display the local coordinates at the center of the plate.*



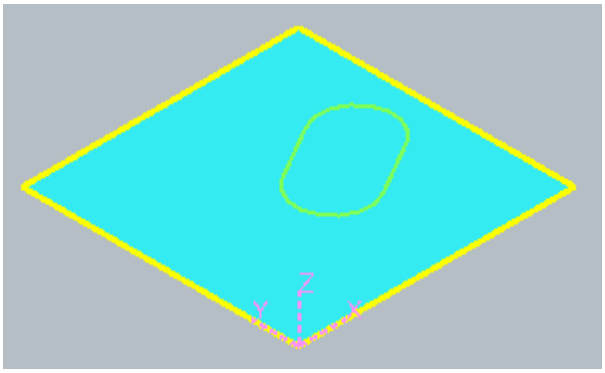
6. If required, select a new shape and new values for the dimension and angle values.

   The preview reflects the new parameters.



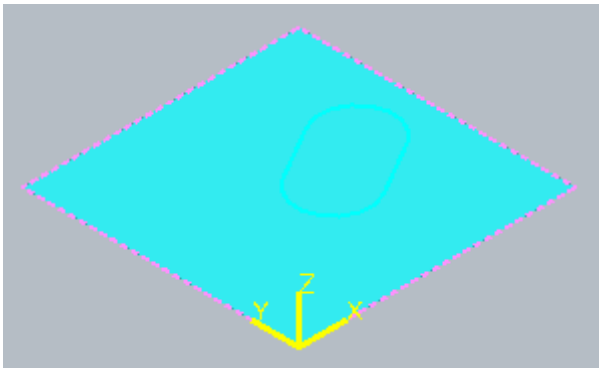
7. If required, type new **X**, **Y**, and **Z** values.

The preview reflects the new opening location.



8. Click **Continue**.

   *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

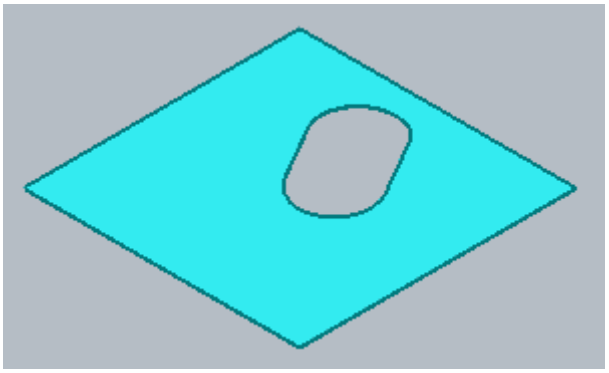
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

9. Click **Close**.

   *The ribbon display returns to the main **Openings** ribbon.*

10. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
11. Click **Finish**.

The software creates the opening.



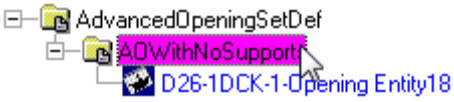
Delete advanced opening geometry

These steps can be performed only when you are creating geometry using an **Advanced Openings** ribbon, before the opening object is created in the model.

1. Click **Select** on the **Advanced Openings** ribbon.

   NOTE Do not click **Select** on the vertical toolbar.

2. In the **Geometric Construction Explorer**, select the opening macro containing the opening geometry to be deleted.



3. Click **Delete** on the ribbon.

TIPS

- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete an opening object after the object is created in the model, see *Delete an opening* (on page 481).

Change an advanced opening to a different opening definition

1. On the vertical toolbar, click **Select**.
2. In a graphic view or in the Workspace Explorer, select the advanced opening to modify.

   *The opening highlights in the graphic view, the ribbon for the opening displays, and the inputs highlight in the **Workspace Explorer**.*

3. In the **Macro Definition** box, click **More**.

   *The **Select Geometric Construction** dialog box displays. Only the appropriate opening geometric constructions display.*

4. Select the needed opening geometric construction, and then click **OK**.

   *The new selection displays in the **Macro Definition** box with a yellow background. Options with blue backgrounds contain new values suggested by the software. Options with white*

backgrounds retain the values from the previous geometric construction.

5. Make the needed changes to options on the ribbon.

   Changed options have a yellow background.

6. Click **Accept** .

   A preview of the changed opening appears.

7. Click **Finish**.

   The software modifies the opening.

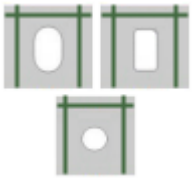
Copy and paste an advanced opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the opening to copy. You can select the opening in a graphic view or from the Workspace Explorer.
4. Click **Edit** > **Copy**.
5. Click **Edit** > **Paste**.
6. Select **Parent of** in the **Paste** dialog box.
7. Select the plate system to receive the opening copy. You can select the plate system in a graphic view or from the **Workspace Explorer**.
8. If needed, select **Select a coordinate system** in the **Paste** dialog box.
9. Select a new coordinate system in the **Workspace Explorer**.
10. Click **OK**.

★ IMPORTANT

- Only the original opening geometry copies. Dimension and support constraints on the original opening geometry are ignored. Use **Copy Similar** to copy constraints. For more information, see *Copy by Family* (on page 1255).
- The copied opening has no relationship to the original opening. Each opening can be modified independently. Use **Copy Similar** if you need a relationship between the openings. For more information, see *Copy by Family* (on page 1255).

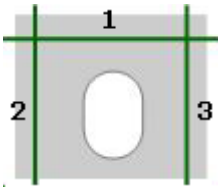
Advanced 3S Opening



Tooltip: AO3SupportOpening

Creates an opening with three supports. The opening is positioned by a dimension from the first support and is centered between the second and third supports.

Select the middle support first for a 3S opening:



You can select the opening shape (circle, oval, or rounded rectangle), and then define the parameters for that shape.

NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

Advanced 3S Opening Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

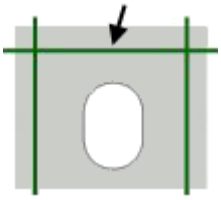
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

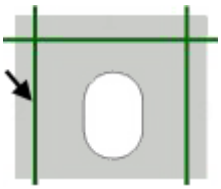
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

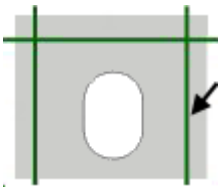
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



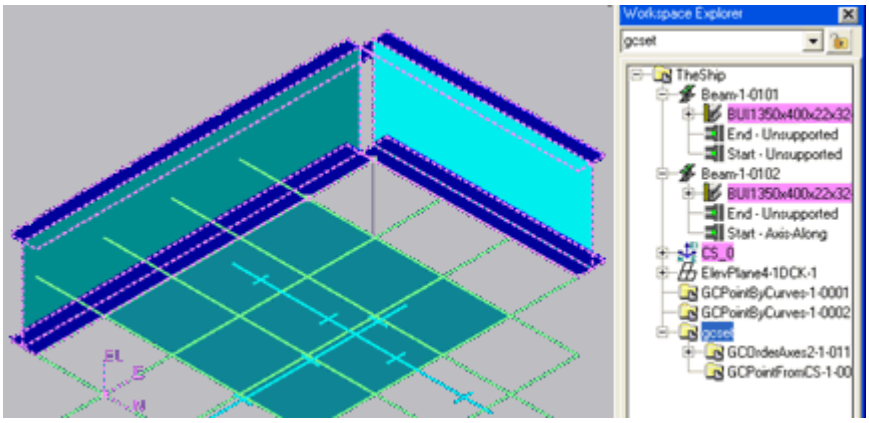
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

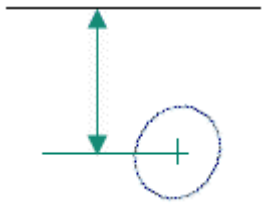
- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

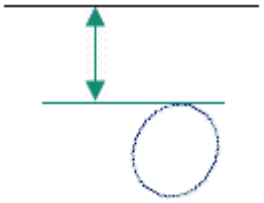
  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



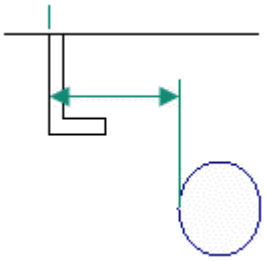
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



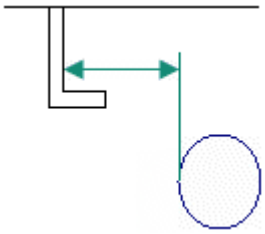
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of these options:Select one of the following options:

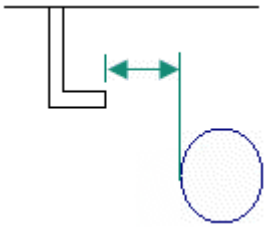
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from he closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. Select **Circle**, **Oval**, or **Rounded Rect**.

Radius - Defines the radius of a circular opening.

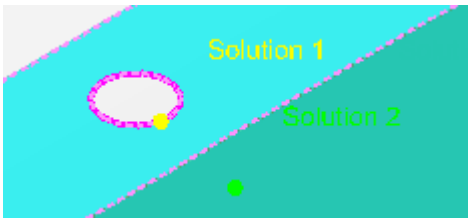
Length - Defines the length of an oval or rounded rectangle opening.

Width - Defines the width of an oval or rounded rectangle opening.

Rounded Rad - Defines the corner radius of a rounded rectangle opening.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place an advanced opening with three supports* (on page 495)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

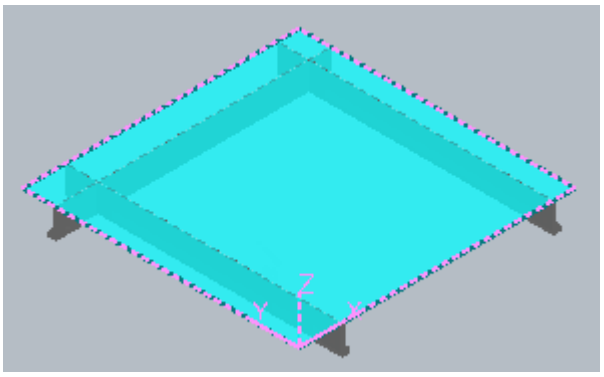
Place an advanced opening with three supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **Custom Shape** tab, click **AO3SupportOpening**.

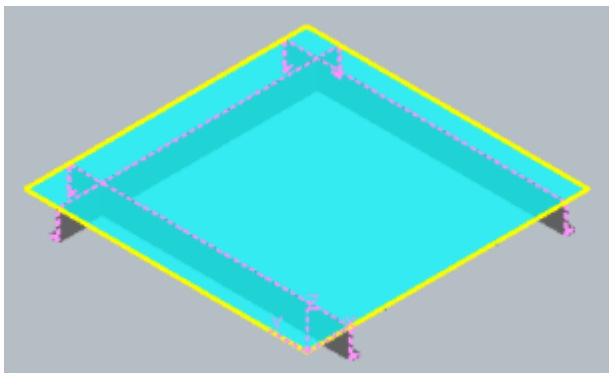
   The ribbon for the macro displays.
5. Select a coordinate system.

   A local coordinate system at the plate origin displays.

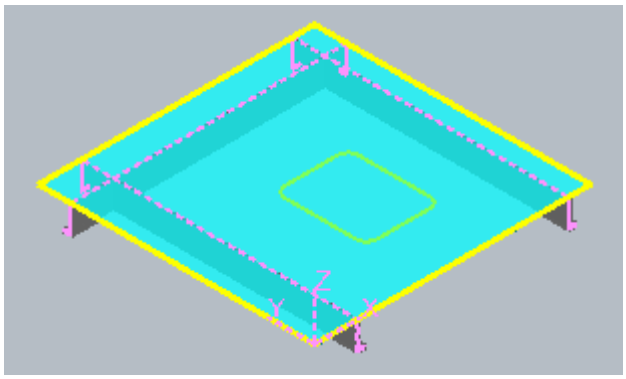


6. Click **2**.

7. Select the middle support, and then select the two side supports.



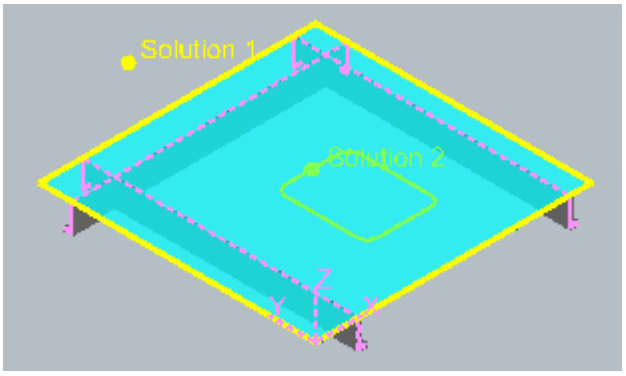
*If the default solution for **Distance Dir** defines geometry on the plate, then a preview displays.*



NOTE *If the default solution for **Distance Dir** does not define geometry on the plate, then a message displays: "Make sure ProjDirection and the DistanceDir are valid and try again."*

8. If a solution is not possible or is not correct:
   a. Click **Distance Dir**.

      The possible solutions display.

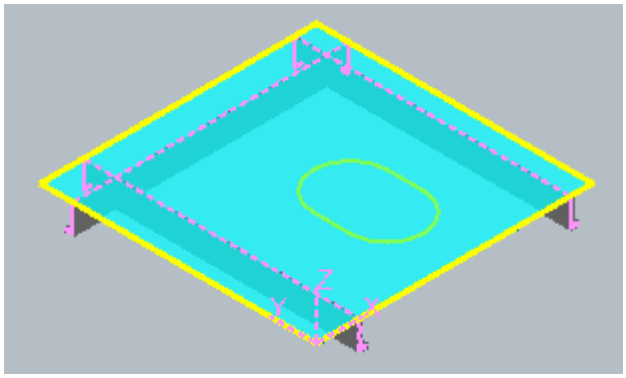


   b. Select the required solution for **Distance Dir**.

      The opening preview displays.

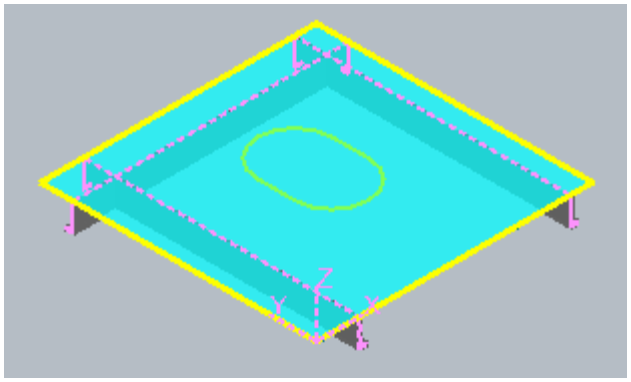
9. If required, select a new shape and new values for the dimension and angle values.

The preview reflects the new parameters.



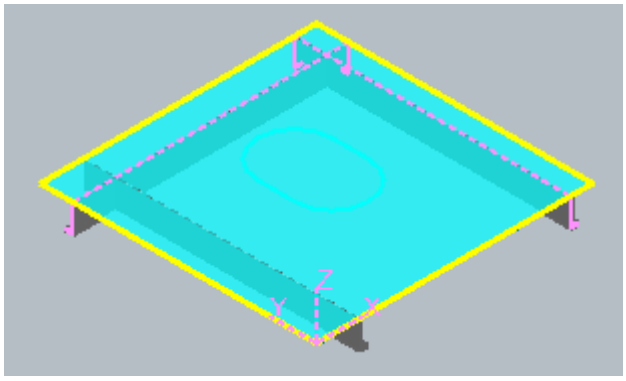
10. If required, change **Distance From** and **Distance** values to change the opening distance with respect to the middle support.

    The preview reflects the new opening location.



11. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

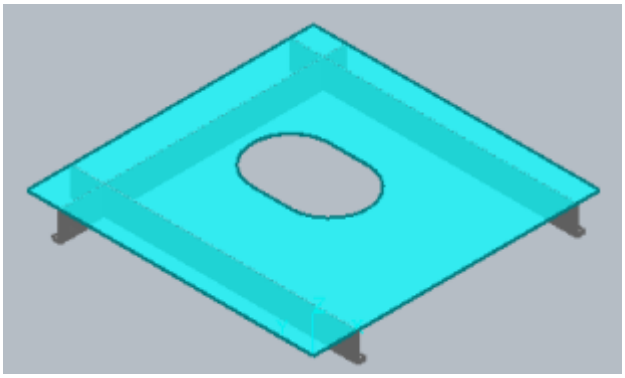
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

12. Click **Close**.

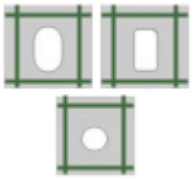
*The ribbon display returns to the main **Openings** ribbon.*

13. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
14. Click **Finish**.

    The software creates the opening.



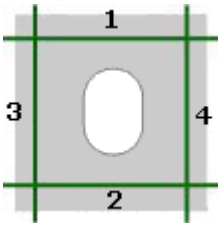
Advanced 4S Opening



Tooltip: AO4SupportOpening

Creates an opening with four supports. The opening is centered between the supports.

Select the supports in order of opposing sets.



You can select the opening shape (circle, oval, or rounded rectangle), and then define the parameters for that shape.

NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

Advanced 4S Opening Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

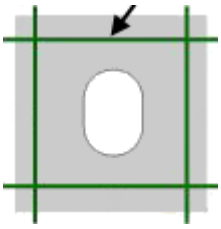
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

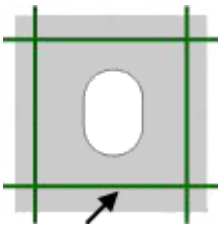
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

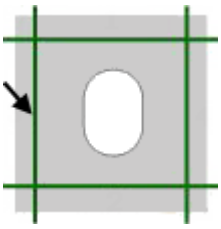
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



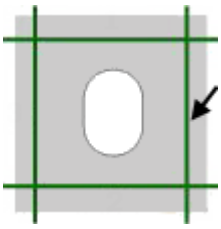
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



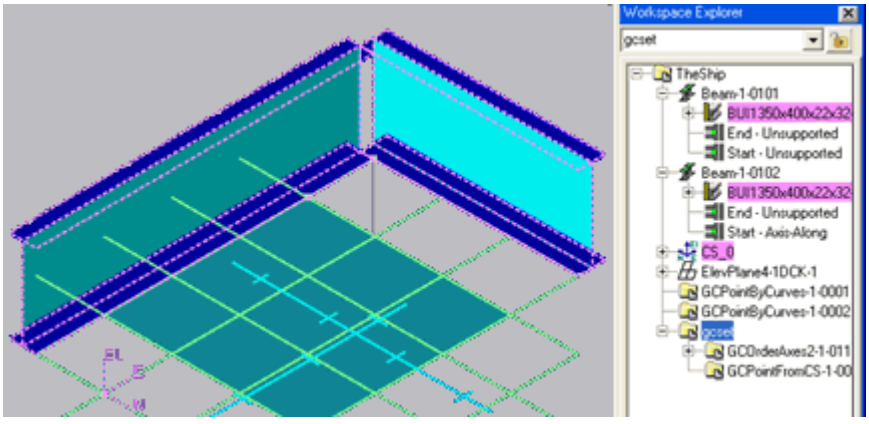
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

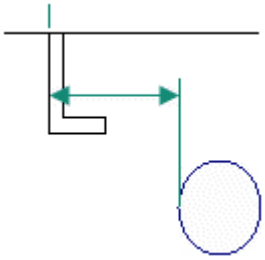
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

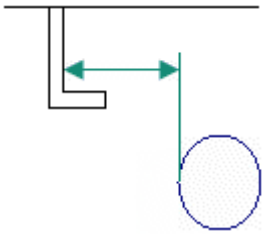
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

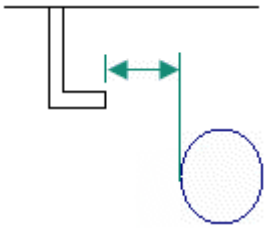
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. Select **Circle**, **Oval**, or **Rounded Rect**. For some opening types, the shape is defined and cannot be changed.

Radius - Defines the radius of a circular opening.

Length - Defines the length of an oval or rounded rectangle opening.

Width - Defines the width of an oval or rounded rectangle opening.

Rounded Rad - Defines the corner radius of a rounded rectangle opening.

What do you want to do?

- *Place an advanced opening with four supports* (on page 502)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

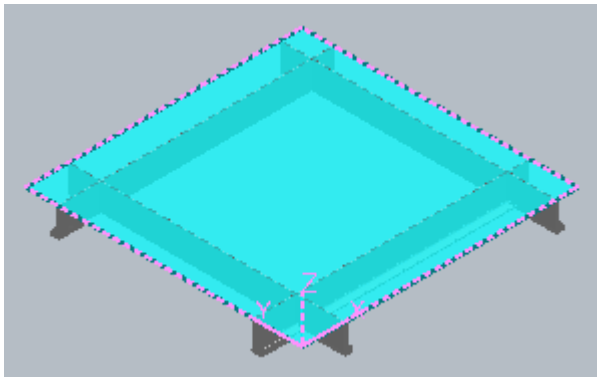
Place an advanced opening with four supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

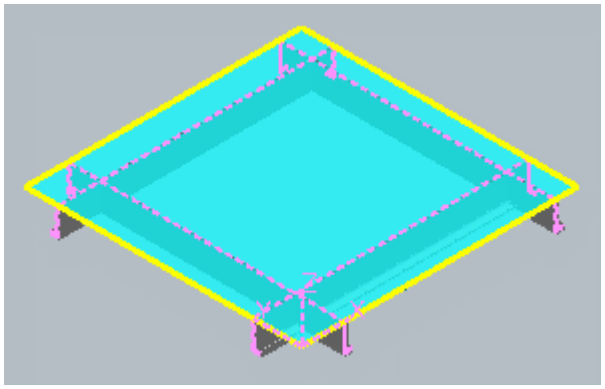
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **Custom Shape** tab, click **AO4SupportOpening**.

   The ribbon for the macro displays.
5. Select a coordinate system.

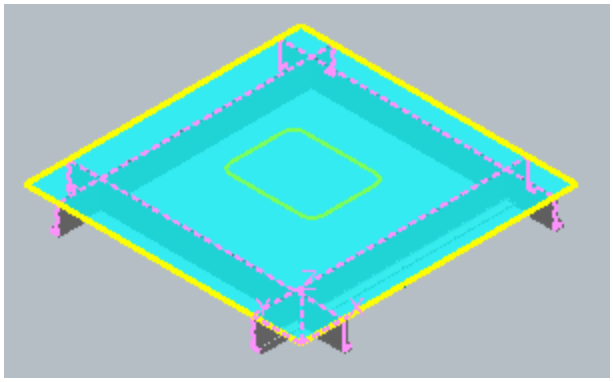
A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the first support, a second support opposite the first support, a third support, and a fourth support opposite the third support.

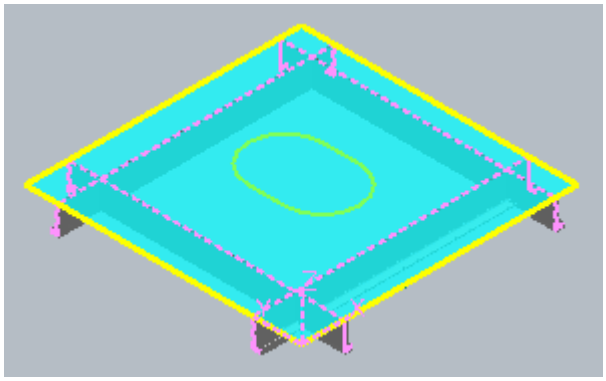


The opening preview displays.



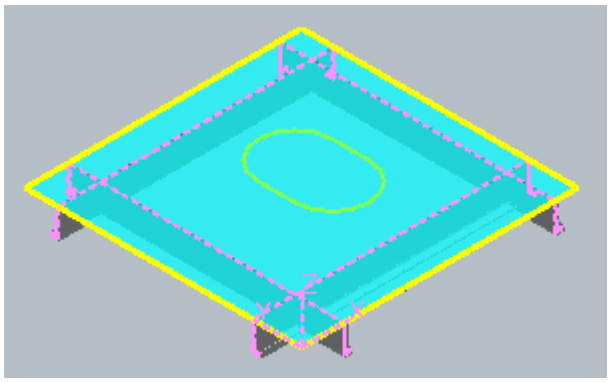
8. If required, select a new shape and new values for the dimension and angle values.

The preview reflects the new parameters.



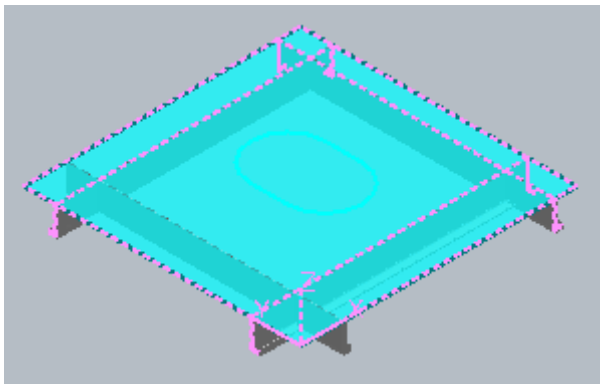
9. If required, change the **Dimension Type** value to change the opening distance from the supports.

   The preview reflects the new opening location.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

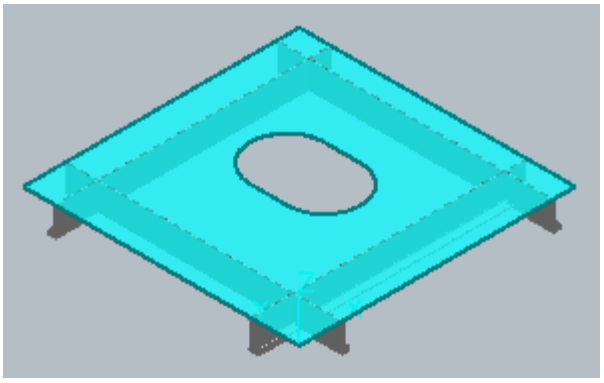
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

*The ribbon display returns to the main **Openings** ribbon.*

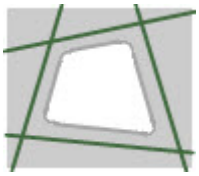
12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

    The software creates the opening.



Support and Offset Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to offset the sides of an opening from the selected supports.

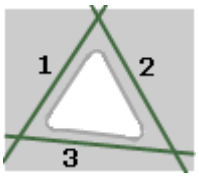
	**3S Support and Offset Opening** - Creates an opening with sides offset from three supports. You can define the offset distance from each support, and the radius at each corner. For more information, see *3S Support and Offset Opening* (on page 505).
	**4S Support and Offset Opening** - Creates an opening with sides offset from four supports. You can define the offset distance from each support, and the radius at each corner. For more information, see *4S Support and Offset Opening* (on page 512).

3S Support and Offset Opening



Tooltip: 3SSupportAndOffsetOpenings

Creates an opening with sides offset from three supports. Select the supports in clockwise or counterclockwise sequential order.



You can define the offset distance from each support, and the radius at each corner.

3S Support and Offset Opening Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.

3 - Specifies the second support used to constrain the opening. This support must intersect the first support. A support is a plate system, profile system, or reference plane.

4 - Specifies the third support used to constrain the opening. This support must intersect the second and first support. A support is a plate system, profile system, or reference plane.

6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

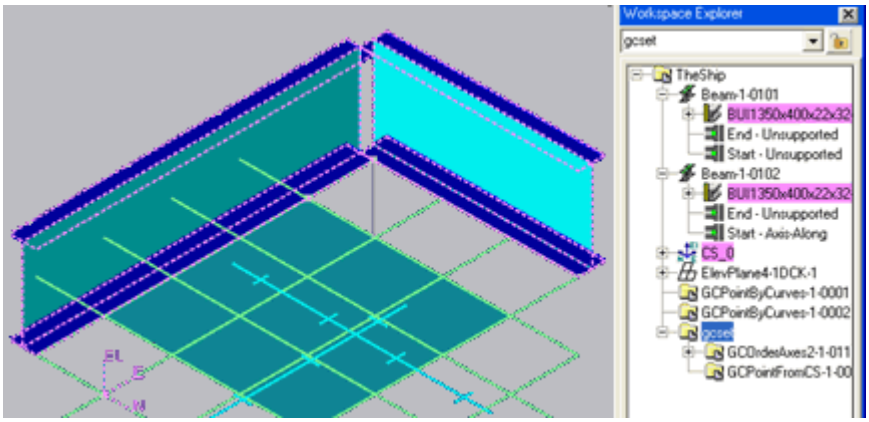
Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.

- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Category - Specifies the type of opening parameters to define. Select **All Supports**, **Support 1**, **Support 2**, **Support 3**, or **Support 4**.

Radius - Defines the radius of each corner of the opening for the **All Supports** category.

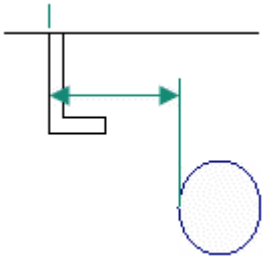
Radius 1, **Radius 2**, **Radius 3**, or **Radius 4** - Defines the corner radius at the start of the support selected for **Category**.

Supp Offset - Defines the offset of the opening from each support for the **All Supports** category.

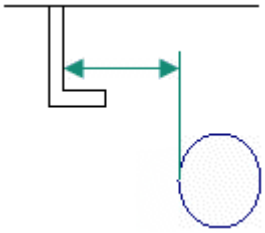
Supp 1 Offset, **Supp 2 Offset**, **Supp 3 Offset**, or **Supp 4 Offset** - Defines the offset of the opening from the support selected for **Category**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support for the **All Supports** category. Select one of the following options:

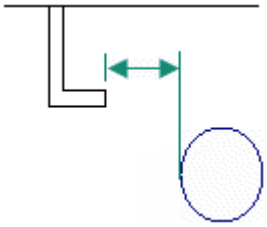
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.

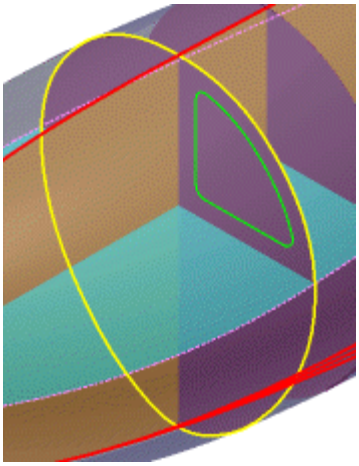


Dimension 1 Type, **Dimension 2 Type**, **Dimension 3 Type**, or **Dimension 4 Type** - Defines how the edge of the opening is located from the support selected for **Category**. See **Dimension Type**, above, for the available options.

Ambiguity 1 - Specifies the location on which to place the opening if two solutions are possible. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. The preview of the opening updates based on the selected value. This option displays only when needed. For more information, see the example below.

Ambiguity 2 - When used in conjunction with **Ambiguity 1**, specifies the location on which to place the opening if four solutions are possible. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. The preview of the opening updates based on the selected values. This option displays only when needed.

In the following example, two plate systems and a circular hull are used as supports and result in four possible solutions. Selecting **+y** for **Ambiguity 1** and **+z** for the **Ambiguity 2** results in the solution shown in green.



NOTE If you select a value that is not valid, such as an x or z value for **Ambiguity 1** or an x value for **Ambiguity 2**, then an error message displays.

What do you want to do?

- *Place an advanced opening offset from three supports* (on page 509)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

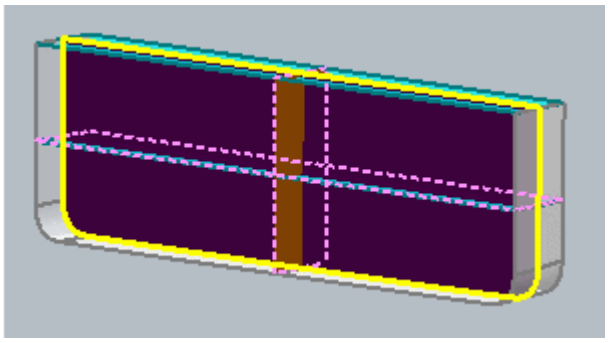
Place an advanced opening offset from three supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **Support and Offset** tab, click **3SSupportAndOffsetOpenings**.

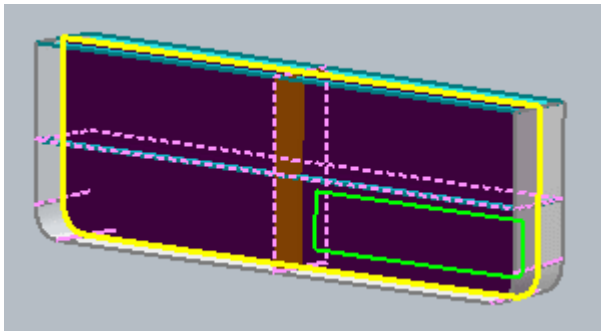
   The ribbon for the macro displays.
5. Select a coordinate system.
6. Click **2**.

7. Select the first support, the second support and the third support in clockwise or counterclockwise sequencial order.

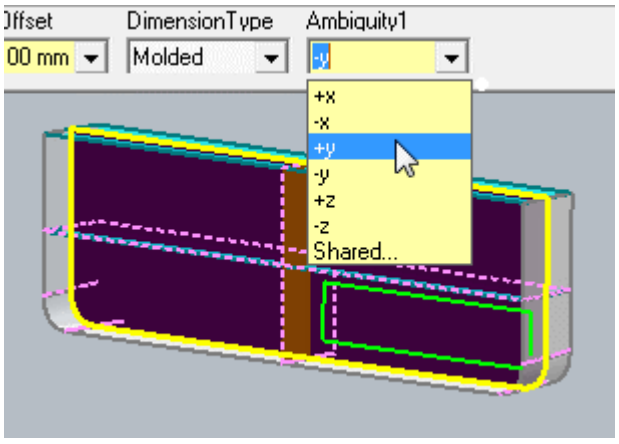


NOTE *In this example, the hull is clipped in the graphic view.*

The opening preview displays.

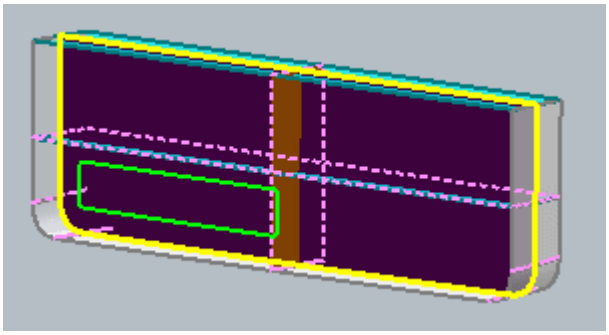


8. If there is more than one possible solution, select the required location.



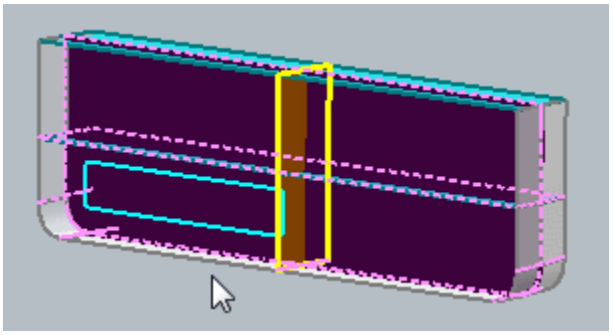
9. If required, change the radius, offset, and dimension type values.

The preview reflects the new opening.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

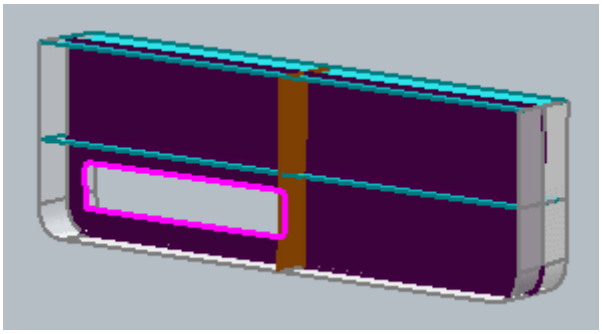
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

    *The ribbon display returns to the main **Openings** ribbon.*

12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

    The software creates the opening.

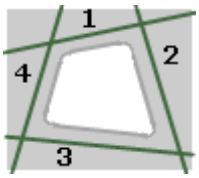


4S Support and Offset Opening



Tooltip: 4SSupportAndOffsetOpenings

Creates an opening with sides offset from four supports. Select the supports in clockwise or counterclockwise sequential order.



You can define the offset distance from each support, and the radius at each corner.

4S Support and Offset Opening Ribbon

What do you want to do?

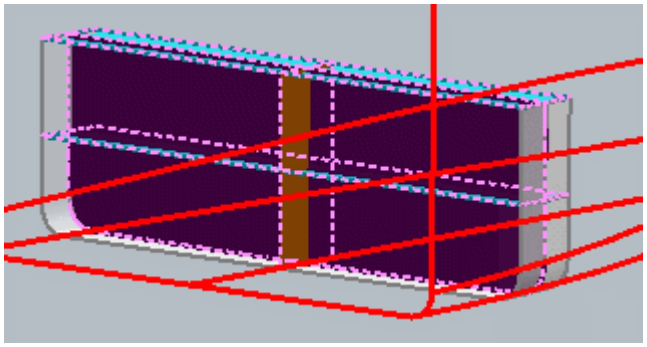
- *Place an advanced opening offset from four supports* (on page 512)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

Place an advanced opening offset from four supports

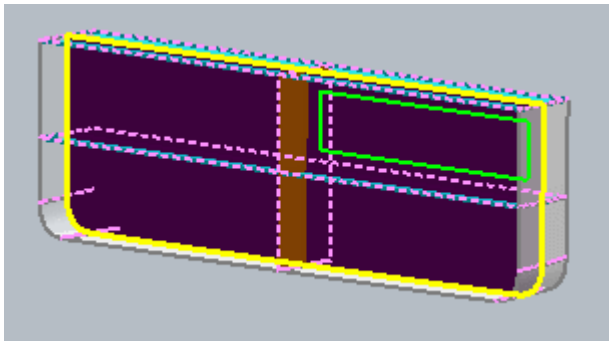
1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **Support and Offset** tab, click **4SSupportAndOffsetOpenings**.

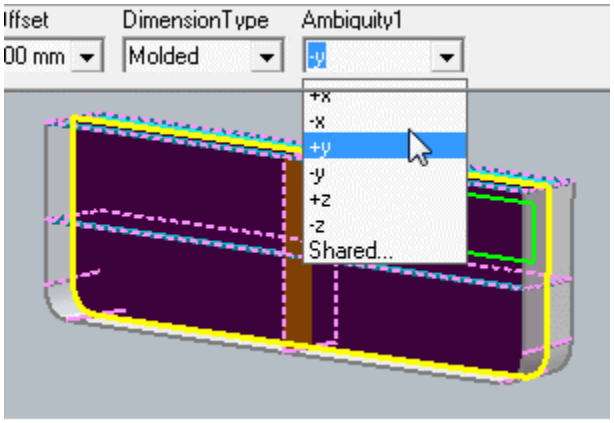
   The ribbon for the macro displays.
5. Select a coordinate system.
6. Click **2**.
7. Select the first support, the second support, the third support, and the fourth support in clockwise or counterclockwise sequencial order.



The opening preview displays.

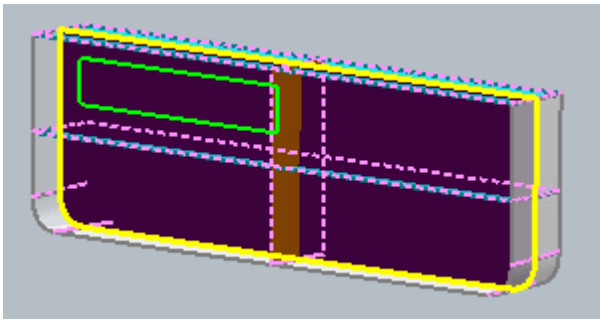


8. If there is more than one possible solution, select the required location.



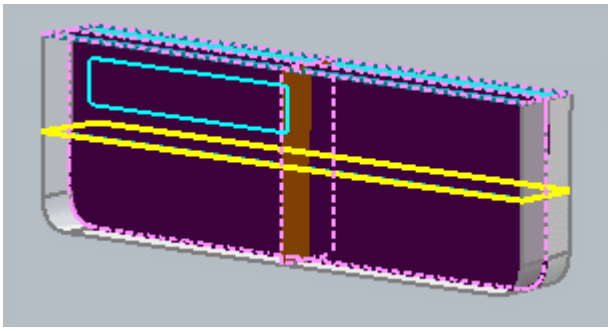
9. If required, change the radius, offset, and dimension type values.

The preview reflects the new opening.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

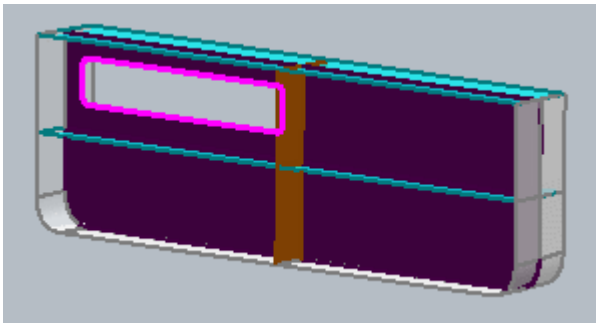
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

    *The ribbon display returns to the main **Openings** ribbon.*

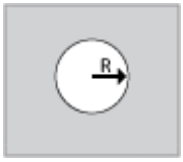
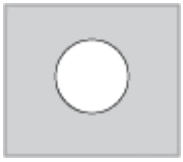
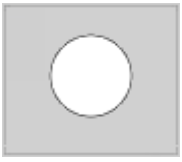
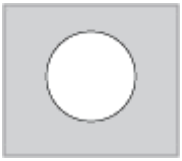
12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

    The software creates the opening.

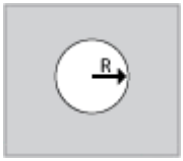


0S Circle Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place circular openings with no supports. The software positions a no-support opening by X-, Y-, and Z-coordinates. The default position is the center of the plate.

	**0S Circle R** - A circular opening with no supports. You can define the radius. For more information, see *Circular Opening with No Supports* (on page 515).
	**0S Circle 200** - A circular opening with no supports. The radius is fixed at 200 mm. For more information, see *Circular Opening with No Supports and a 200 mm Fixed Radius* (on page 520).
	**0S Circle 250** - A circular opening with no supports. The radius is fixed at 250 mm. For more information, see *Circular Opening with No Supports and a 250 mm Fixed Radius* (on page 524).
	**0S Circle 300** - A circular opening with no supports. The radius is fixed at 300 mm. For more information, see *Circular Opening with No Supports and a 300 mm Fixed Radius* (on page 526).

Circular Opening with No Supports



Tooltip: 0SCircle R

Creates a circular opening with no supports that you position with X-, Y-, and Z-coordinates. The default position is the center of the plate. You can define the radius of the opening.

0S Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

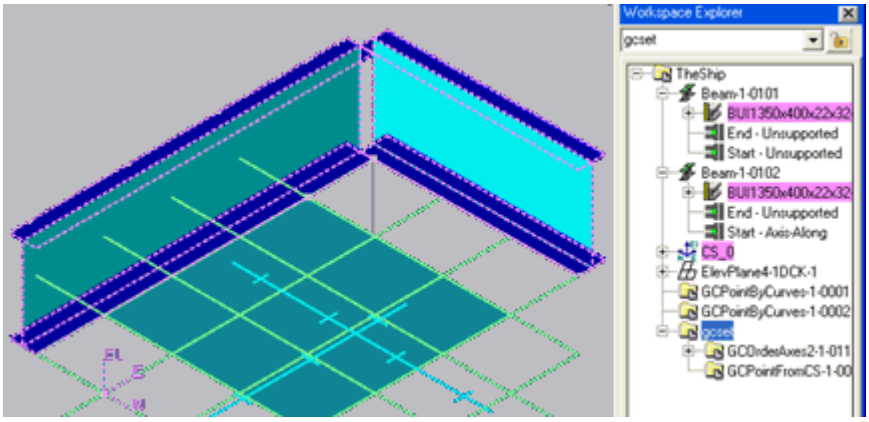
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of the circular opening.

What do you want to do?

- *Place a circular opening with no supports* (on page 518)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

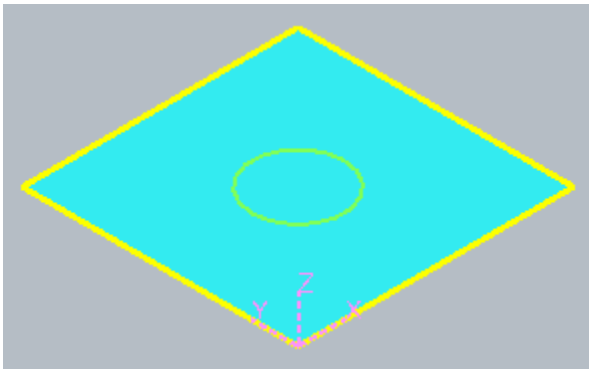
Place a circular opening with no supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **0S Circle** tab, click **0SCircle R**.

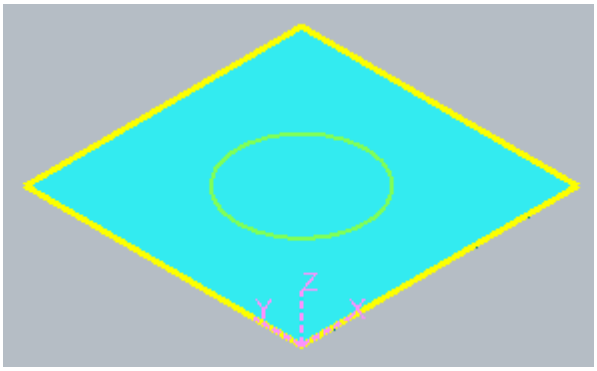
   The ribbon for the macro displays.
5. Select a coordinate system.

   *A local coordinate system at the plate origin displays. A preview of the opening using the default radius displays at the center of the plate. **X**, **Y**, and **Z** display the local coordinates at the center of the plate.*



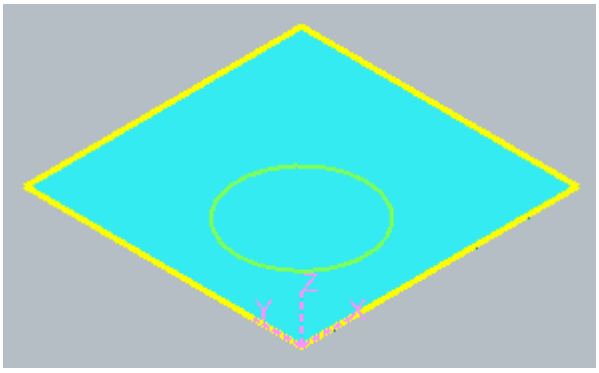
6. If required, type a new value for **Radius**.

   The preview reflects the new opening size.



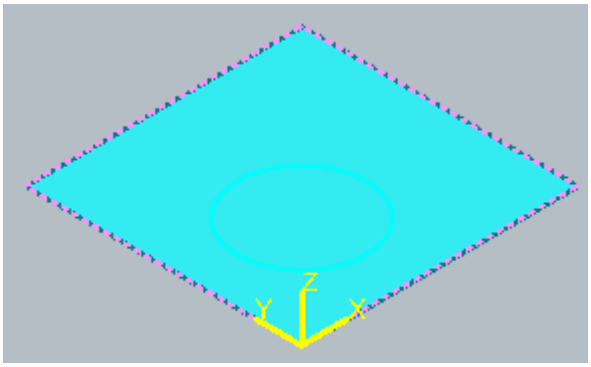
7. If required, type new **X**, **Y**, and **Z** values.

The preview reflects the new opening location.



8. Click **Continue**.

   *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

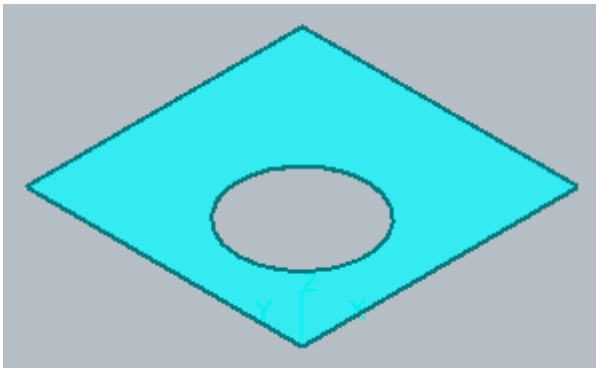
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

9. Click **Close**.

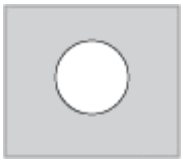
   *The ribbon display returns to the main **Openings** ribbon.*

10. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
11. Click **Finish**.

The software creates the opening.



Circular Opening with No Supports and a 200 mm Fixed Radius



Tooltip: 0SCircle 200

Creates a circular opening with a 200 mm radius and no supports. The opening is positioned by X, Y, and Z coordinates. The default position is the center of the plate.

0S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

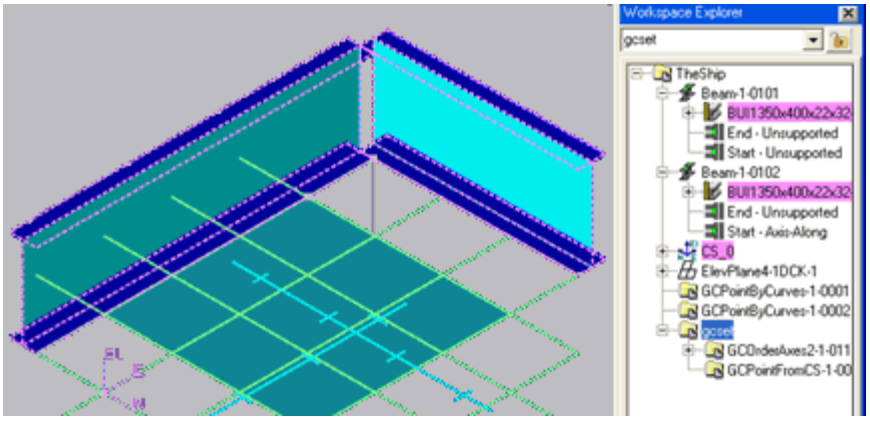
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of the circular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

- *Place a circular opening with no supports and a fixed radius* (on page 522)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

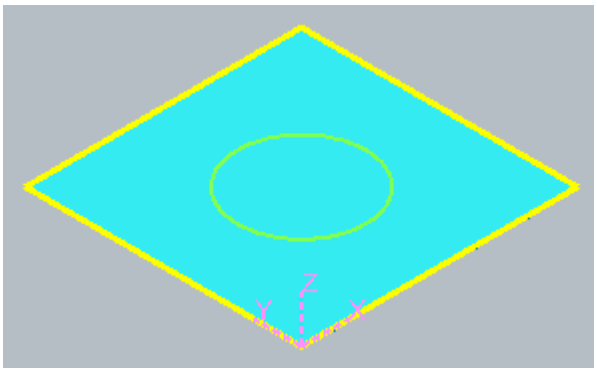
Place a circular opening with no supports and a fixed radius

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection** .

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **0S Circle** tab, click **0SCircle 200**, **0SCircle 250**, or **0SCircle 300**.

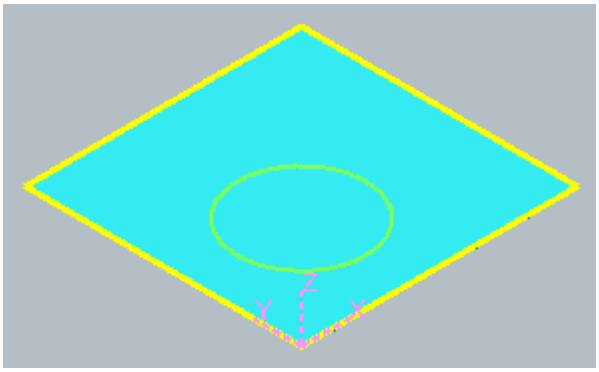
   The ribbon for the macro displays.
5. Select a coordinate system.

*A local coordinate system at the plate origin displays. A preview of the opening using the fixed radius displays at the center of the plate. **X**, **Y**, and **Z** display the local coordinates at the center of the plate.*



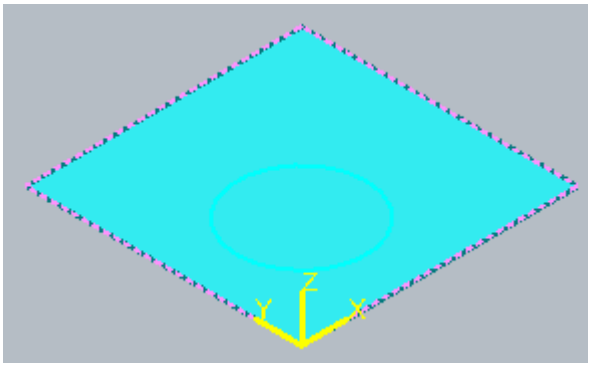
6. If required, type new **X**, **Y**, and **Z** values.

   The preview reflects the new opening location.



7. Click **Continue**.

   *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

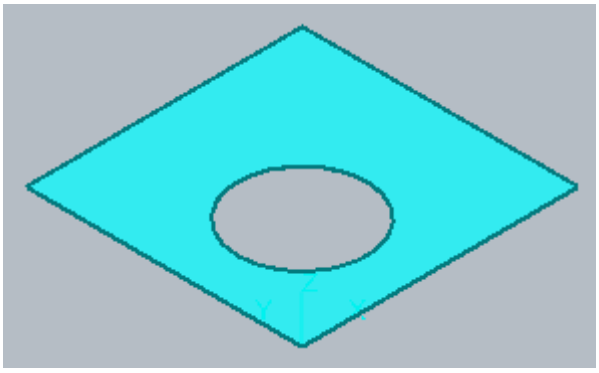
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

8. Click **Close**.

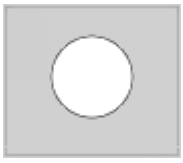
*The ribbon display returns to the main **Openings** ribbon.*

9. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
10. Click **Finish**.

    The software creates the opening.



Circular Opening with No Supports and a 250 mm Fixed Radius



Tooltip: 0SCircle 250

Creates a circular opening with a 250 mm radius and no supports. The opening is positioned by X-, Y-, and Z-coordinates. The default position is the center of the plate.

0S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.

- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

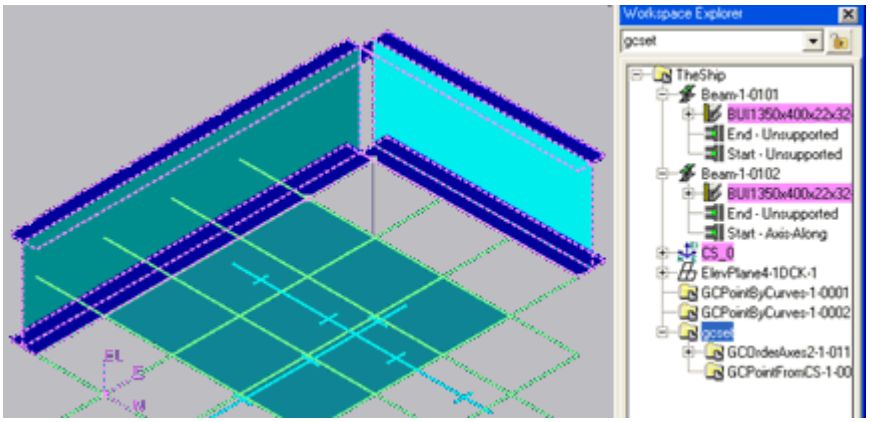
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**. This option is not used for a circular opening.

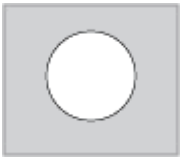
Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of the circular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

- *Place a circular opening with no supports and a fixed radius* (on page 522)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

Circular Opening with No Supports and a 300 mm Fixed Radius



Tooltip: 0SCircle 300

Creates a circular opening with a 300 mm radius and no supports. The opening is positioned by X, Y, and Z coordinates. The default position is the center of the plate.

0S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

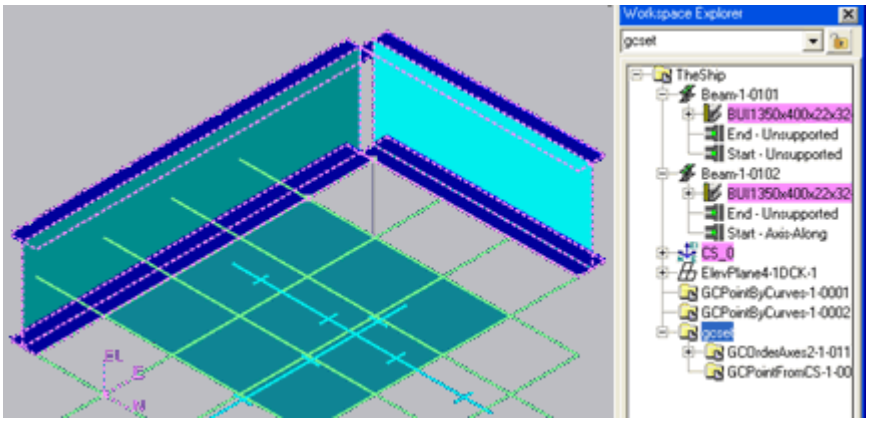
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.

- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

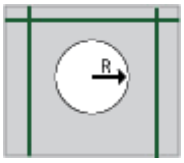
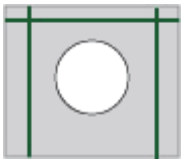
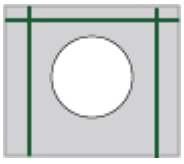
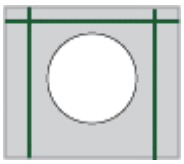
Radius - Defines the radius of the circular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

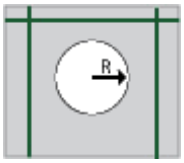
- *Place a circular opening with no supports and a fixed radius* (on page 522)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

3S Circle Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place circular openings with three supports. The software positions a three-support opening by dimensions from the first support and is centers the opening between the second and third supports.

	**3S Circle R** - Creates a circular opening with three supports. You can define the radius. For more information, see *Circular Opening with Three Supports* (on page 530).
	**3S Circle 200** - Creates a circular opening with three supports. The radius is fixed at 200 mm. For more information, see *Circular Opening with Three Supports and a 200 mm Fixed Radius* (on page 538).
	**3S Circle 250** - Creates a circular opening with three supports. The radius is fixed at 250 mm. For more information, see *Circular Opening with Three Supports and a 250 mm Fixed Radius* (on page 545).
	**3S Circle 300** - Creates a circular opening with three supports. The radius is fixed at 300 mm. For more information, see *Circular Opening with Three Supports and a 300 mm Fixed Radius* (on page 550).

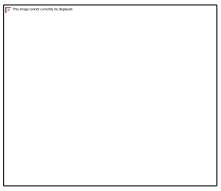
Circular Opening with Three Supports



Tooltip: Circle RxD

Creates a circular opening with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports. You define the radius of the opening.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

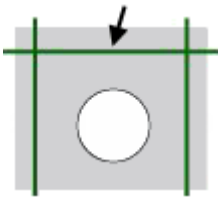
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

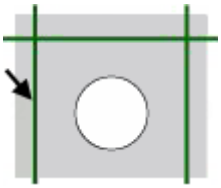
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

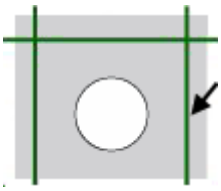
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



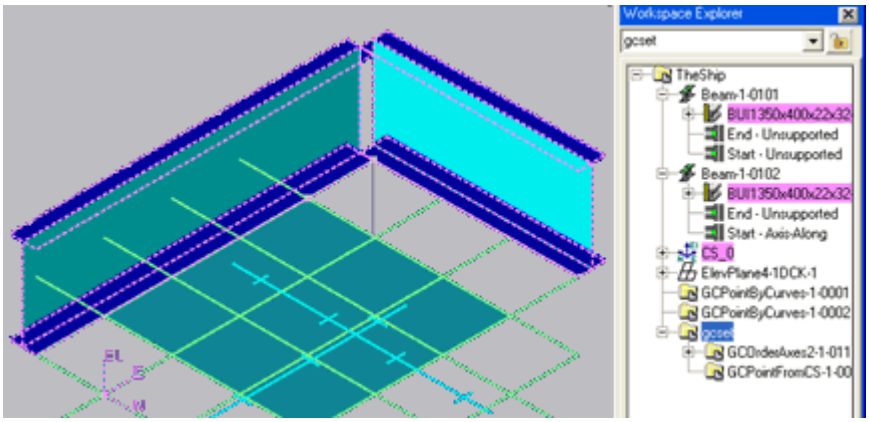
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

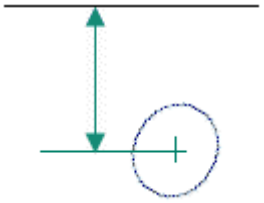
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

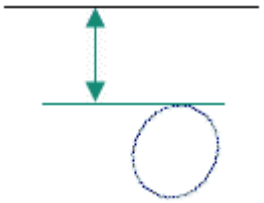
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



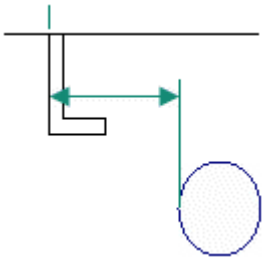
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



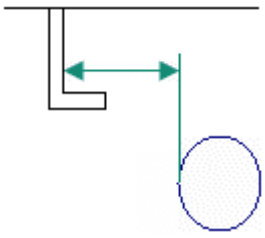
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

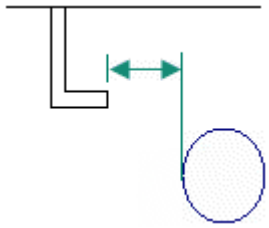
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

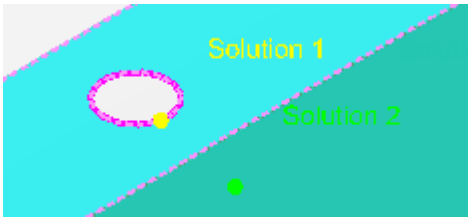
Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of the circular opening.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place a circular opening with three supports* (on page 534)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

Place a circular opening with three supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.

3. Click **Define by catalog selection** .

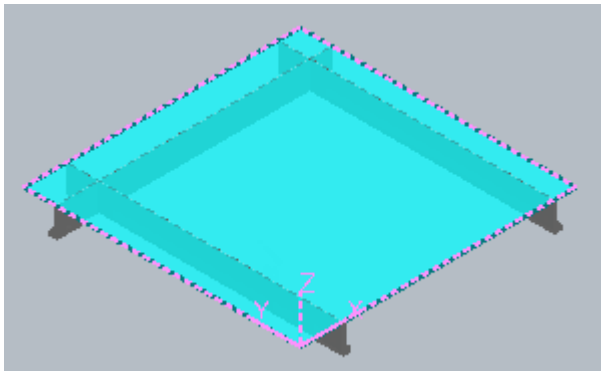
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*

4. On the **3S Circle** tab, click **Circle RxD**.

   The ribbon for the macro displays.

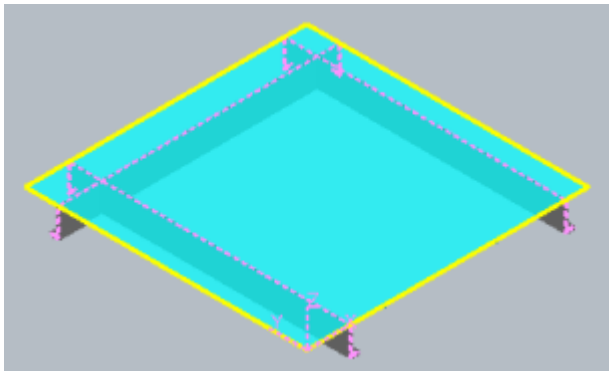
5. Select a coordinate system.

   A local coordinate system at the plate origin displays.

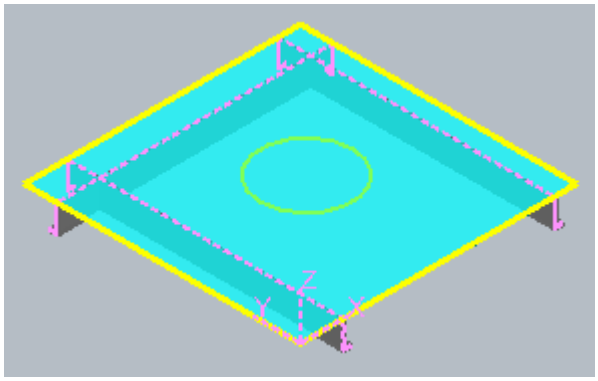


6. Click **2**.

7. Select the middle support, and then select the two side supports.

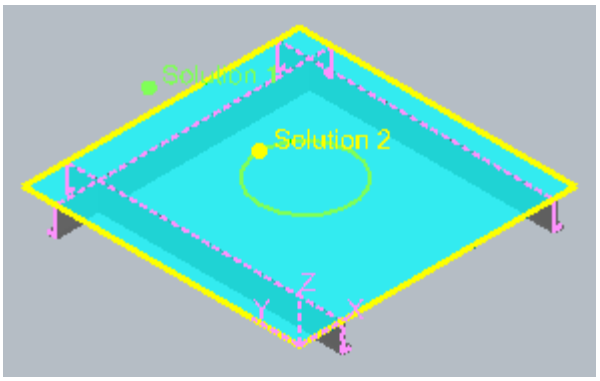


   *If the default solution for **Distance Dir** defines geometry on the plate, a preview displays.*

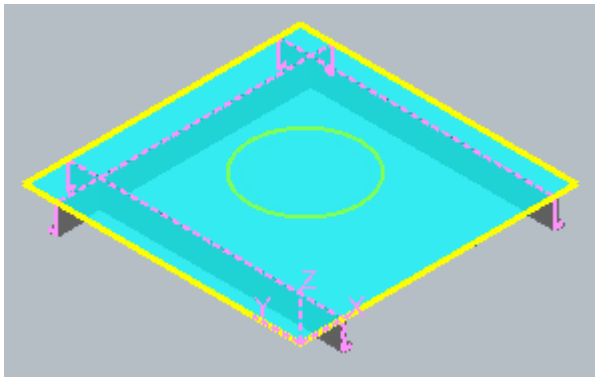


NOTE *If the default solution for **Distance Dir** does not define geometry on the plate, then a message displays:*
"Make sure ProjDirection and the DistanceDir are valid and try again."

8. If a solution is not possible or is not correct:
   a. Click **Distance Dir**.
      The possible solutions display.

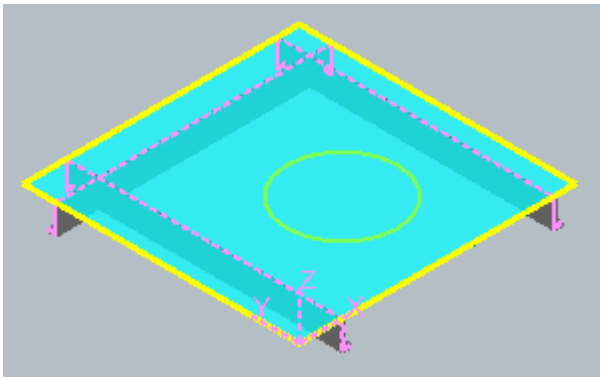


   b. Select the required solution for **Distance Dir**.
      The opening preview displays.
9. If required, type a new value for the **Radius**.
   The preview reflects the opening size.



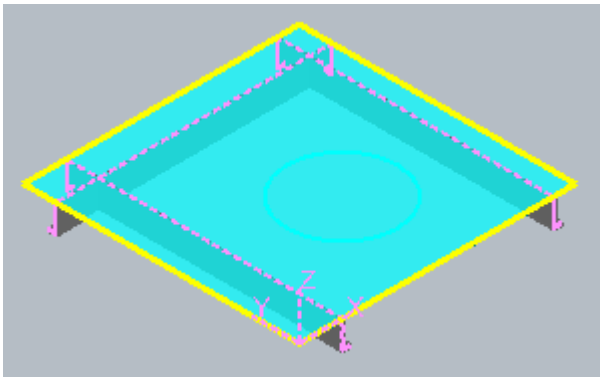
10. If required, change **Distance From** and **Distance** values to change the opening distance with respect to the middle support.

The preview reflects the new opening location.



11. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

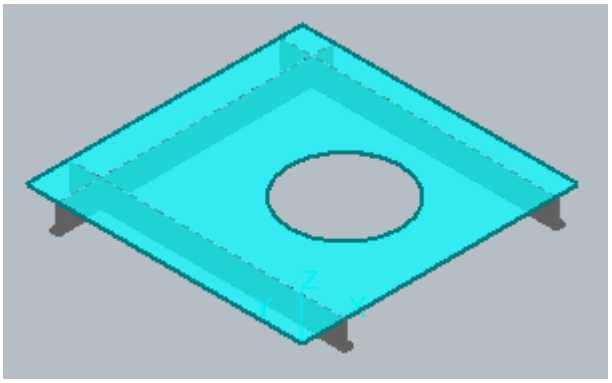
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

12. Click **Close**.

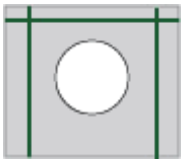
    *The ribbon display returns to the main **Openings** ribbon.*

13. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
14. Click **Finish**.

The software creates the opening.



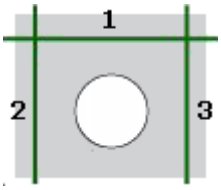
Circular Opening with Three Supports and a 200 mm Fixed Radius



Tooltip: Circle 200

Creates a circular opening with a 200 mm radius and three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

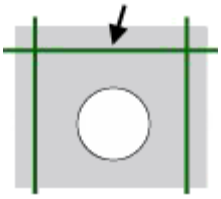
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

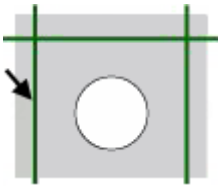
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

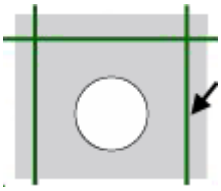
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



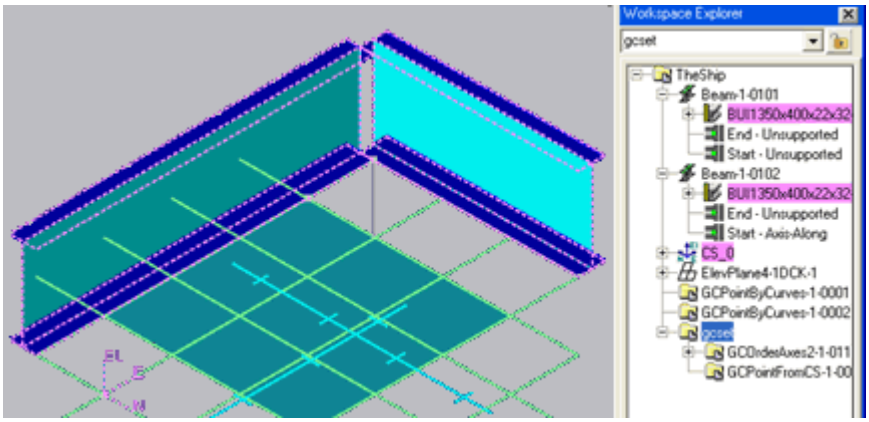
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

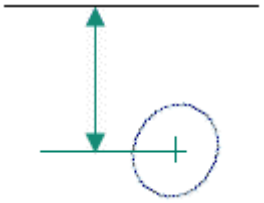
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

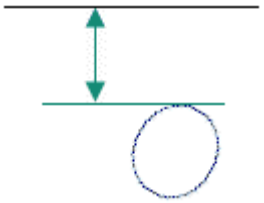
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



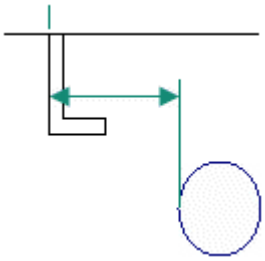
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



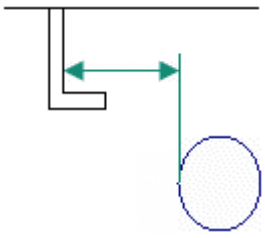
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

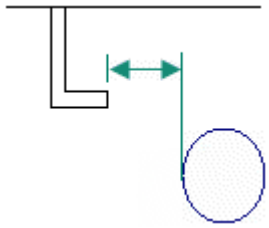
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

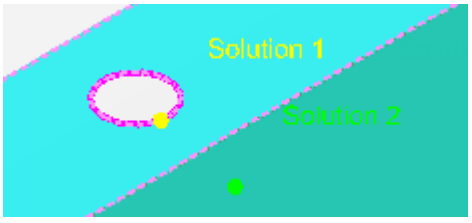
Angle - Defines the rotation angle of the opening about the center of the opening The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of a circular opening. This option is unavailable and set to the fixed radius of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place a circular opening with three supports and a fixed radius* (on page 543)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

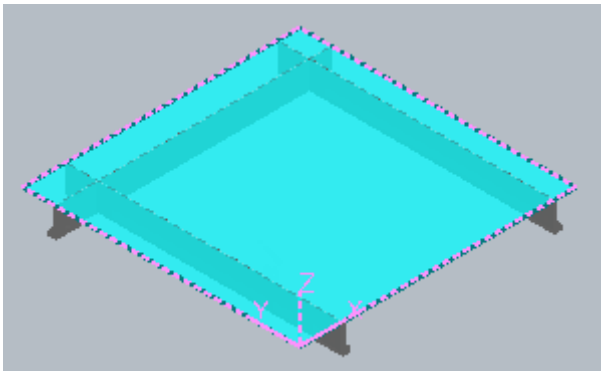
Place a circular opening with three supports and a fixed radius

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

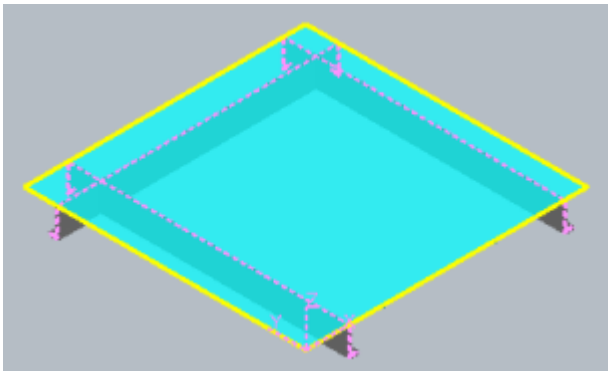
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **3S Circle** tab, click **Circle 200**, **Circle 250**, or **Circle 300**.

   The ribbon for the macro displays.
5. Select a coordinate system.

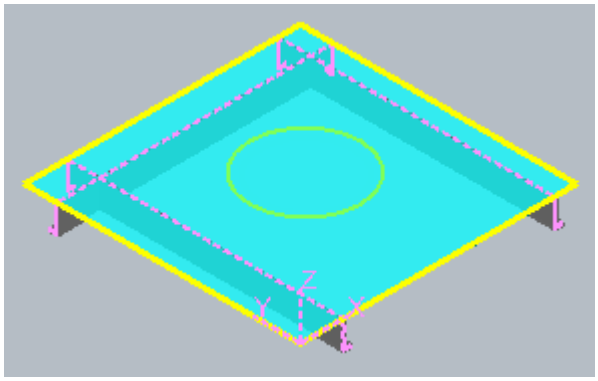
   A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the middle support, and then select the two side supports.



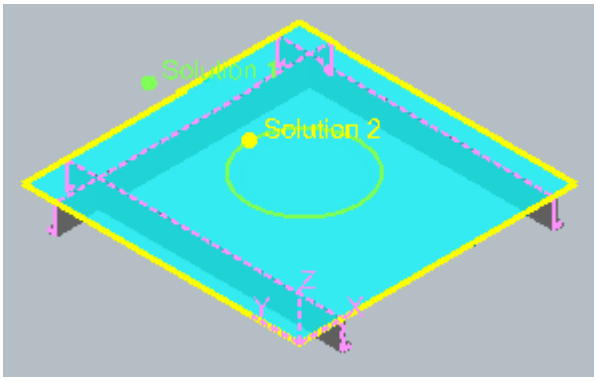
*If the default solution for **Distance Dir** defines geometry on the plate, a preview displays.*



NOTE *If the default solution for **Distance Dir** does not define geometry on the plate, then a message displays: "Make sure ProjDirection and the DistanceDir are valid and try again."*

8. If a solution is not possible or is not correct:
   a. Click **Distance Dir**.

      The possible solutions display.

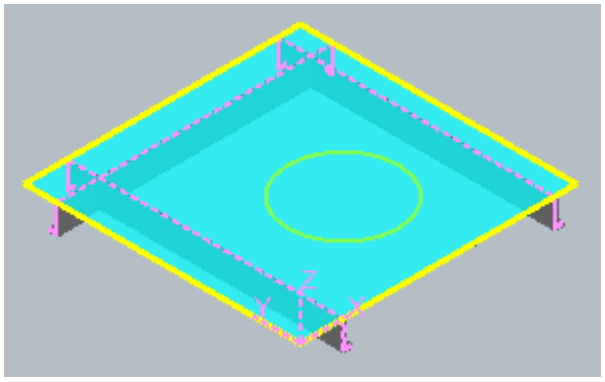


   b. Select the required solution for **Distance Dir**.

      The opening preview displays.

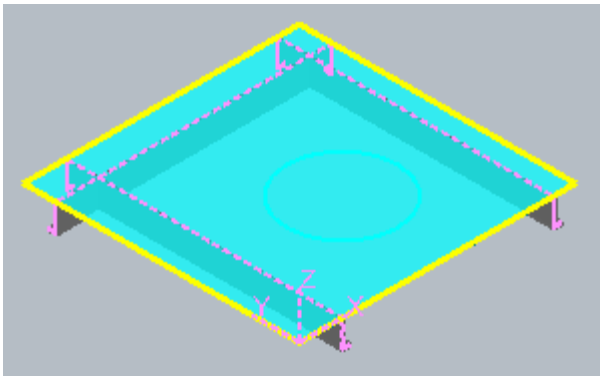
9. If required, change **Distance From** and **Distance** values to change the opening distance with respect to the middle support.

   The preview reflects the new opening location.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

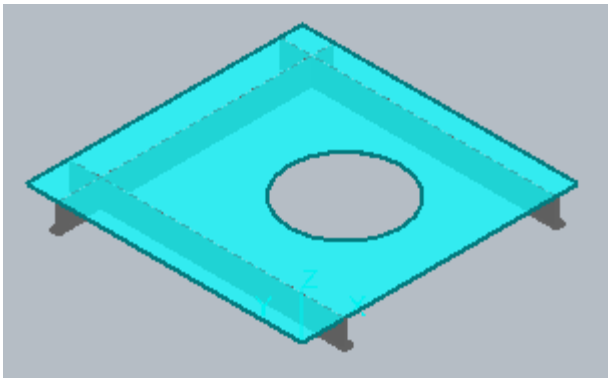
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

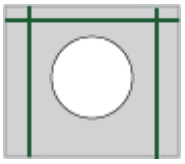
    *The ribbon display returns to the main **Openings** ribbon.*

12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

    The software creates the opening.



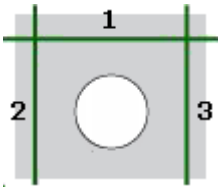
Circular Opening with Three Supports and a 250 mm Fixed Radius



Tooltip: Circle 250

Creates a circular opening with a 250 mm radius and three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

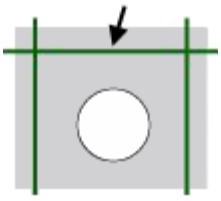
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

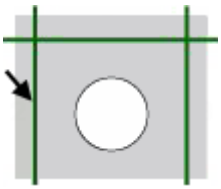
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

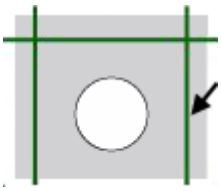
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



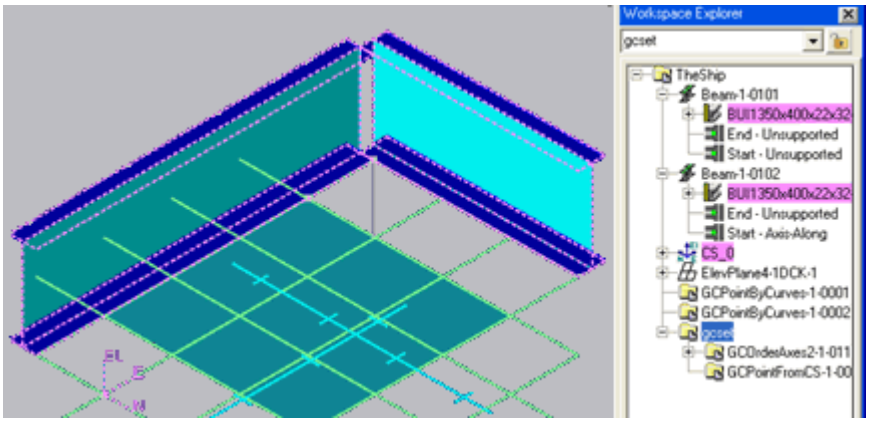
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

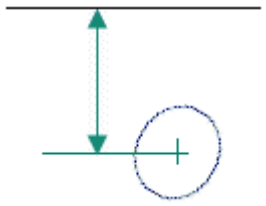
- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

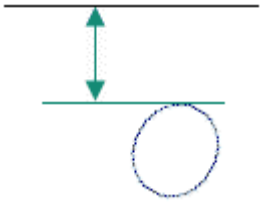
  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



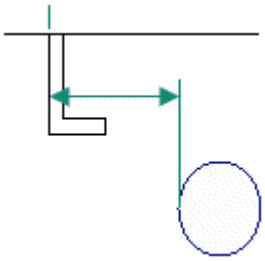
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



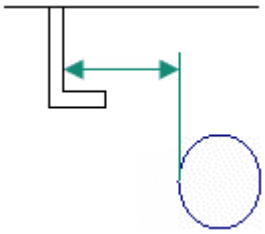
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

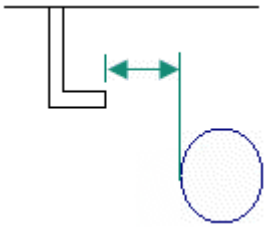
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

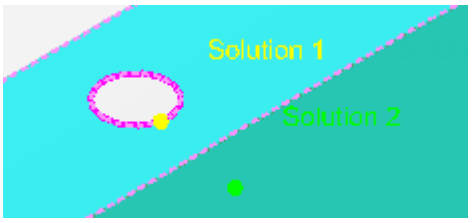
Angle - Defines the rotation angle of the opening about the center of the opening The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of a circular opening. This option is unavailable and set to the fixed radius of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

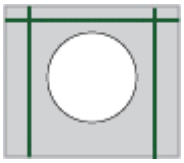
Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place a circular opening with three supports and a fixed radius* (on page 543)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

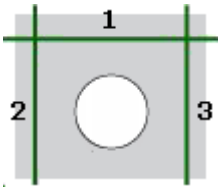
Circular Opening with Three Supports and a 300 mm Fixed Radius



Tooltip: Circle 300

Creates a circular opening with a 300 mm radius and three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

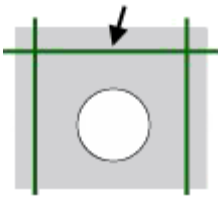
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

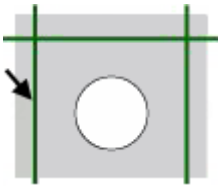
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

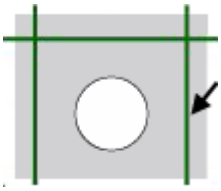
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



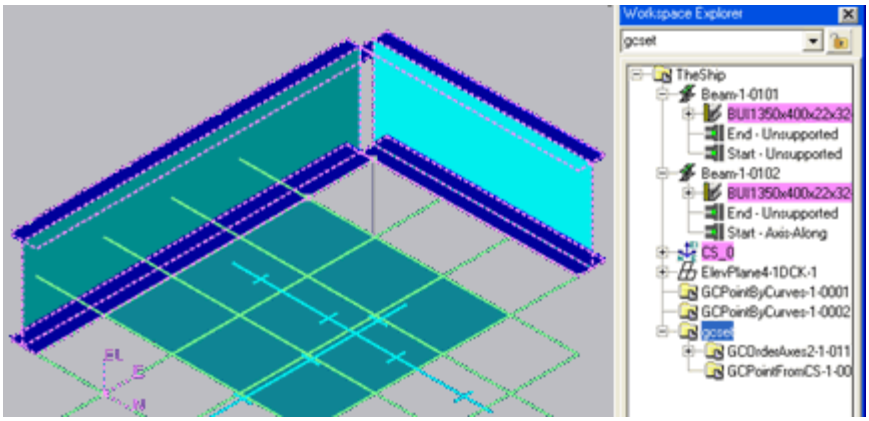
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

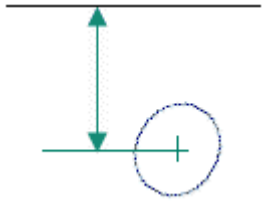
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

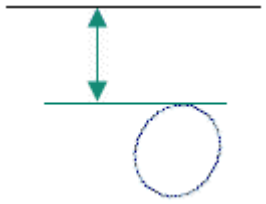
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



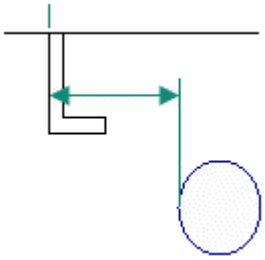
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



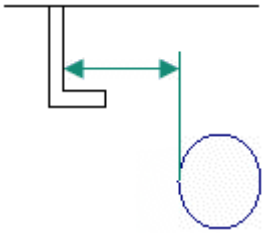
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

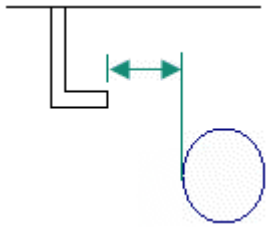
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

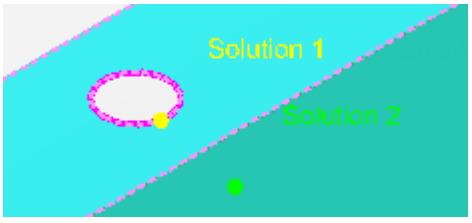
Angle - Defines the rotation angle of the opening about the center of the opening The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of a circular opening. This option is unavailable and set to the fixed radius of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.

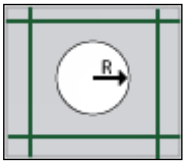
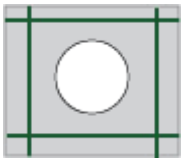
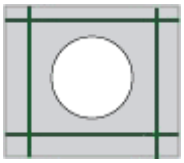
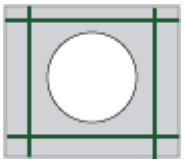


What do you want to do?

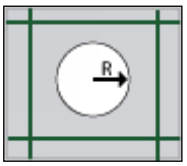
- *Place a circular opening with three supports and a fixed radius* (on page 543)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

4S Circle Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place circular openings with four supports. The software centers a four-support opening between opposing supports.

	**4S Circle R** - Creates a circular opening with four supports. You can define the radius. For more information, see *Circular Opening with Four Supports* (on page 555).
	**4S Circle 200** - Creates a circular opening with four supports. The radius is fixed at 200 mm. For more information, see *Circular Opening with Four Supports and a 200 mm Fixed Radius* (on page 562).
	**4S Circle 250** - Creates a circular opening with four supports. The radius is fixed at 250 mm. For more information, see *Circular Opening with Four Supports and a 250 mm Fixed Radius* (on page 568).
	**4S Circle 300** - Creates a circular opening with four supports. The radius is fixed at 300 mm. For more information, see *Circular Opening with Four Supports and a 300 mm Fixed Radius* (see "*Circular Opening with Four Supports and a 250 mm Fixed Radius*" on page 572).

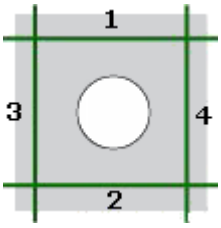
Circular Opening with Four Supports



Tooltip: 4SCircle R

Creates a circular opening with four supports. The software centers the opening between opposing supports. You define the radius of the opening.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

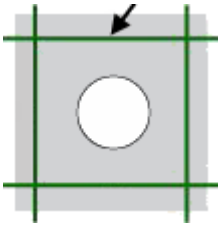
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

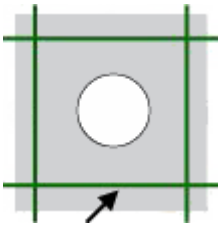
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

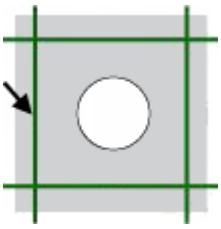
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



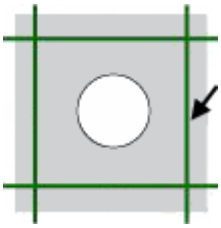
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



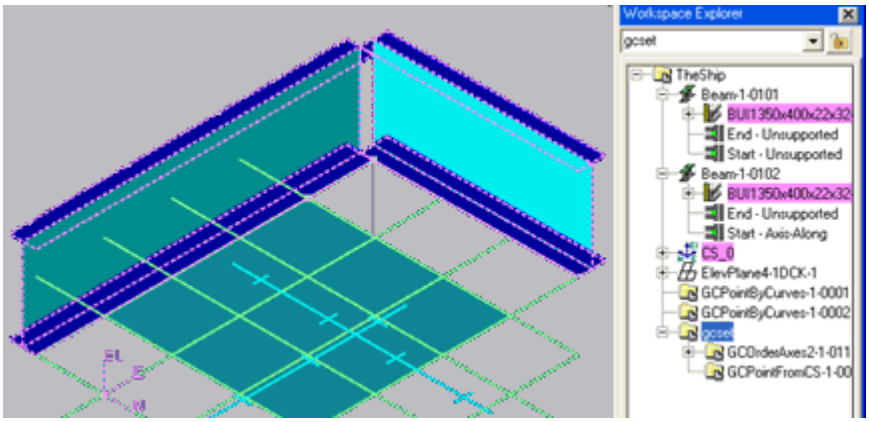
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

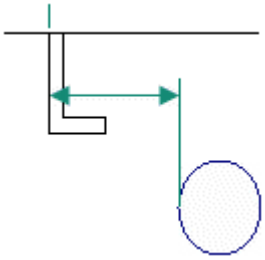
- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

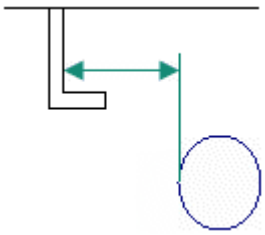
  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

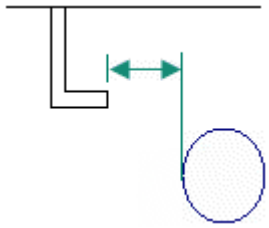
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of the circular opening.

What do you want to do?

- *Place a circular opening with four supports* (on page 559)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

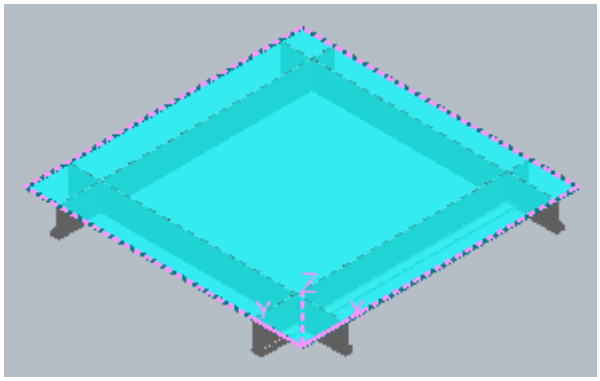
Place a circular opening with four supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

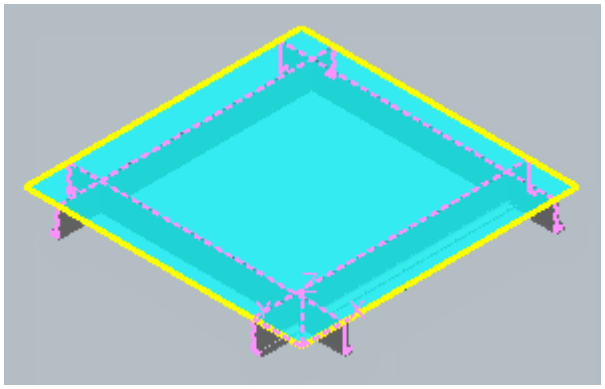
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **4S Circle** tab, click **4SCircle R**.

   The ribbon for the macro displays.
5. Select a coordinate system.

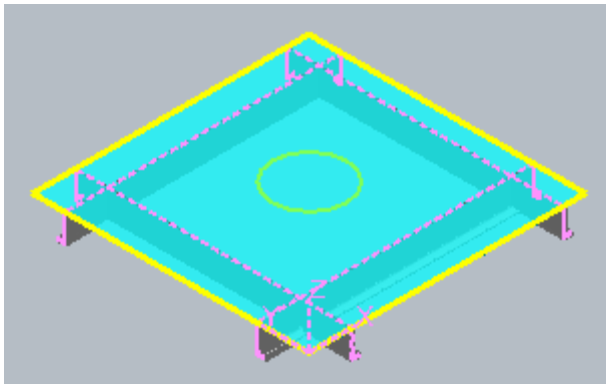
A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the first support, a second support opposite the first support, a third support, and a fourth support opposite the third support.

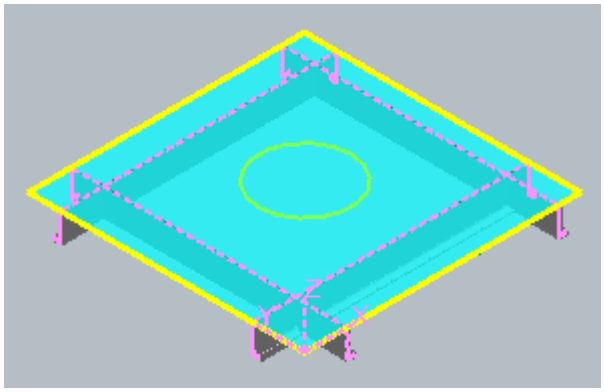


The opening preview displays.



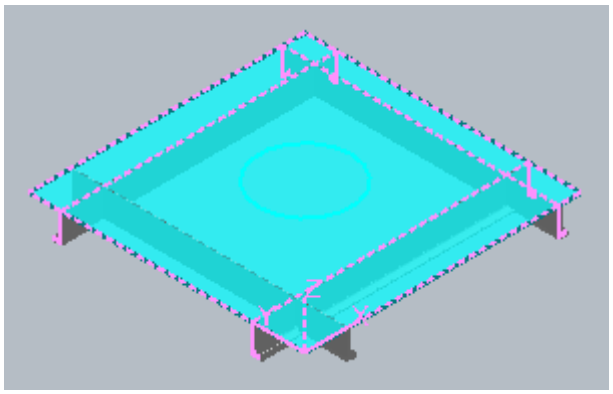
8. If required, type a new value for **Radius**.

The preview reflects the new size.



9. Click **Continue**.

   *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

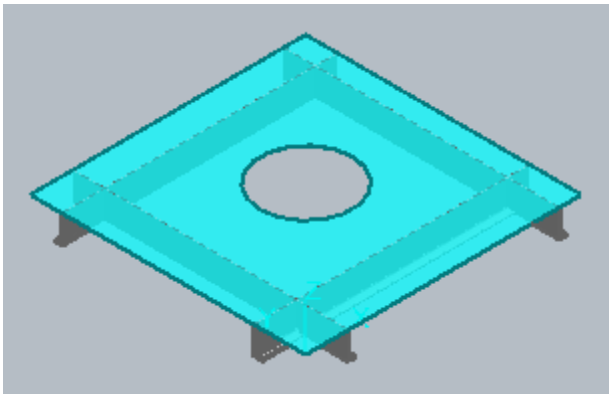
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

10. Click **Close**.

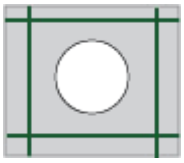
    *The ribbon display returns to the main **Openings** ribbon.*

11. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
12. Click **Finish**.

The software creates the opening.



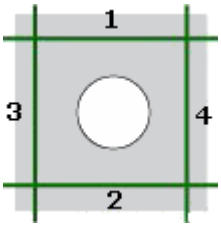
Circular Opening with Four Supports and a 200 mm Fixed Radius



Tooltip: 4SCircle 200

Creates a circular opening with a 200 mm radius and four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

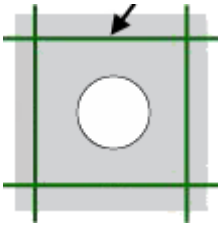
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

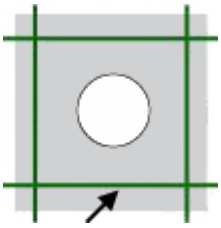
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

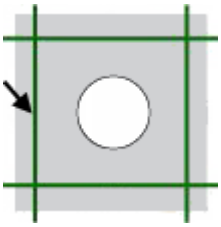
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



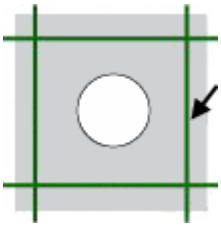
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



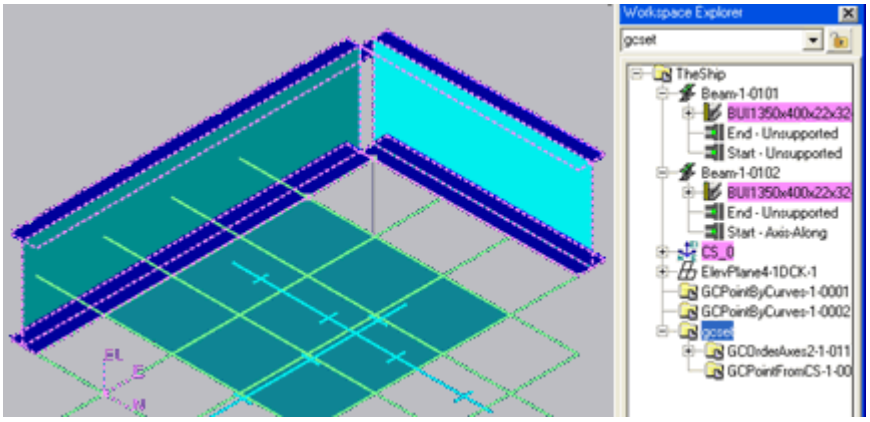
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

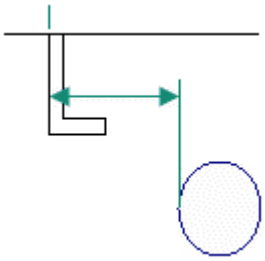
"The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and does not display when editing an object.

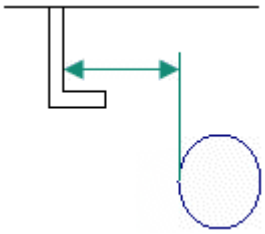
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

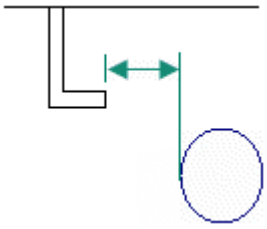
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of the circular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

- *Place a circular opening with four supports and a fixed radius* (on page 566)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

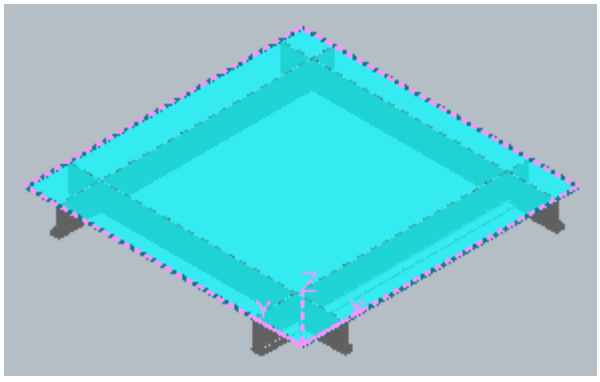
Place a circular opening with four supports and a fixed radius

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **4S Circle** tab, click **4SCircle 200**, **4SCircle 250**, or **4SCircle 300**.

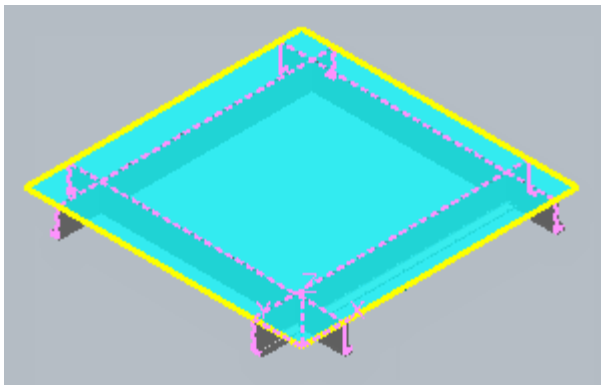
   The ribbon for the macro displays.
5. Select a coordinate system.

   A local coordinate system at the plate origin displays.

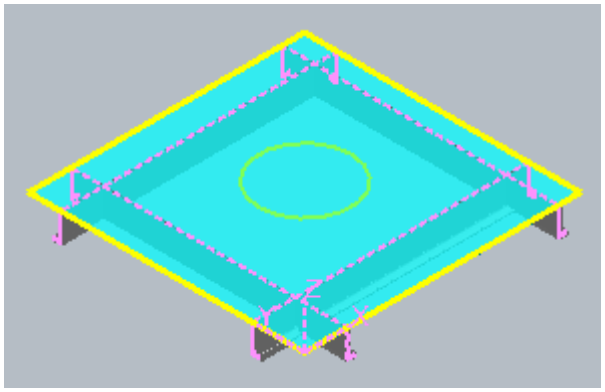


6. Click **2**.

7. Select the first support, a second support opposite the first support, a third support, and a fourth support opposite the third support.

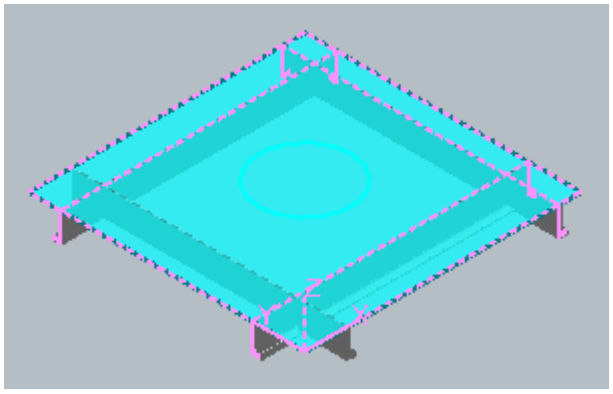


The opening preview displays.



8. Click **Continue**.

*The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

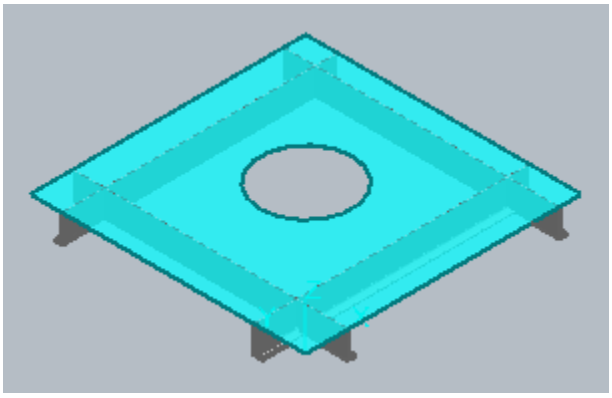
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

9. Click **Close**.

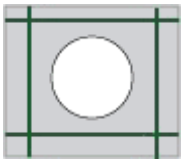
   *The ribbon display returns to the main **Openings** ribbon.*

10. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
11. Click **Finish**.

    The software creates the opening.



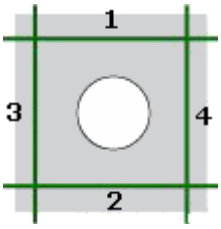
Circular Opening with Four Supports and a 250 mm Fixed Radius



Tooltip: 4SCircle 250

Creates a circular opening with a 250 mm radius and four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

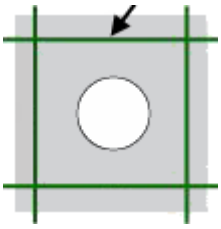
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

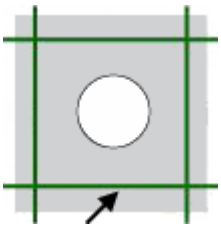
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

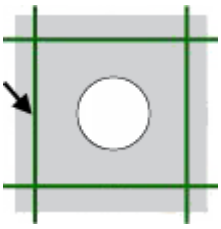
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



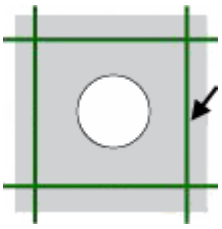
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



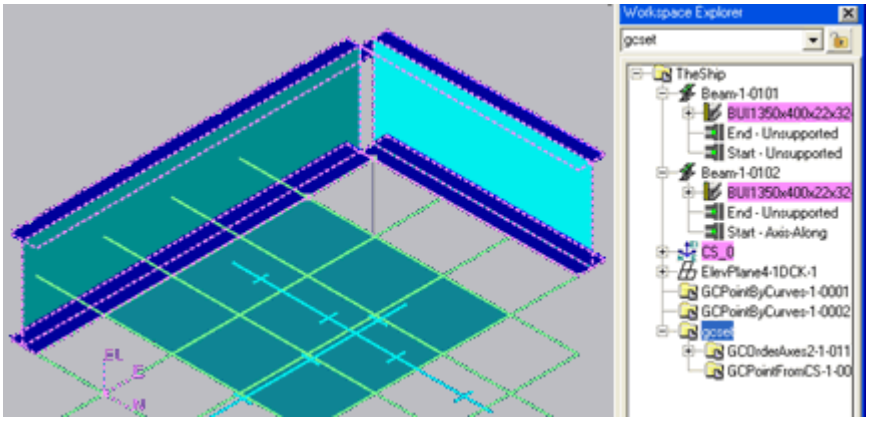
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

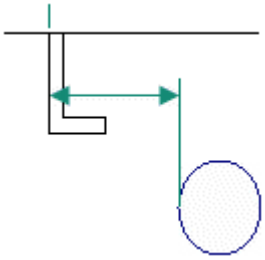
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

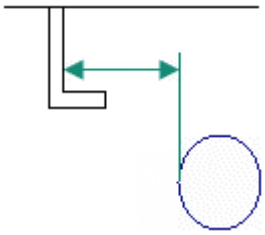
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

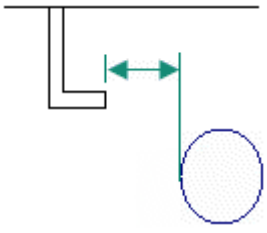
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**. This option is not used for a circular opening.

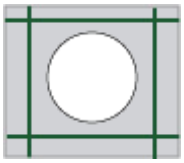
Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

Radius - Defines the radius of the circular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

- *Place a circular opening with four supports and a fixed radius* (on page 566)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

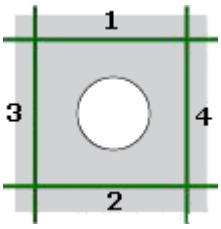
Circular Opening with Four Supports and a 250 mm Fixed Radius



Tooltip: 4SCircle 300

Creates a circular opening with a 300 mm radius and four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Radius Circle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

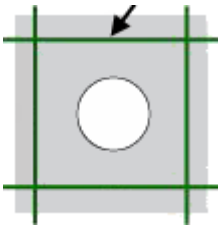
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

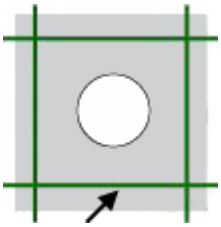
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

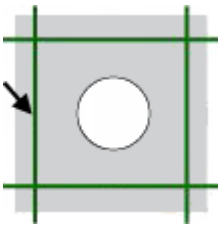
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



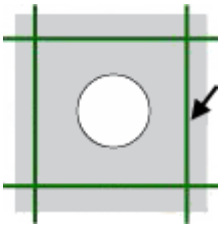
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



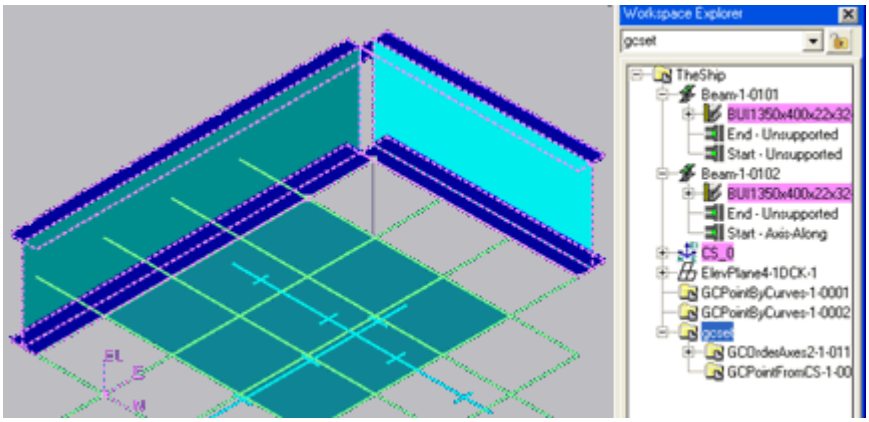
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

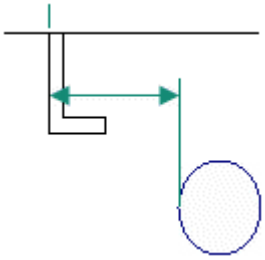
- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

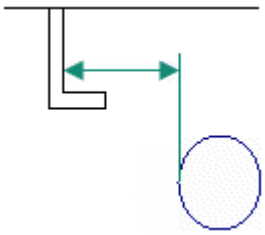
  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

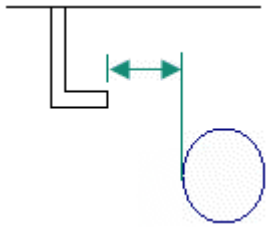
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**. This option is not used for a circular opening.

Shape - Defines the shape of the opening. This option is unavailable and set to **Circle**.

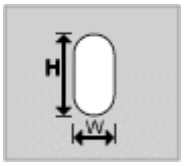
Radius - Defines the radius of the circular opening. This option is unavailable and set to the fixed radius of the macro.

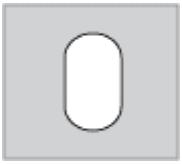
What do you want to do?

- *Place a circular opening with four supports and a fixed radius* (on page 566)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

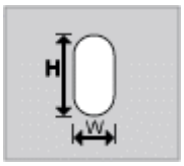
0S Oval Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place oval openings with no supports. The software positions a no-support opening by X-, Y-, and Z-coordinates. The default position is the center of the plate.

	**0S Oval HxW** - Creates an oval opening with no supports. You can define the length and the width. For more information, see *Oval Opening with No Supports* (on page 577).

	**0S Oval 500x250** - Creates a 500 mm length and 250 mm width oval opening with no supports. For more information, see *Oval Opening with No Supports, a 500 mm Length, and a 250 mm Width* (on page 582).
	**0S Oval 850x500** - Creates an 850 mm length and 500 mm width oval opening with no supports. For more information, see *Oval Opening with No Supports, an 850 mm Length, and a 500 mm Width* (on page 586).

Oval Opening with No Supports



Tooltip: 0SOval HxW

Creates an oval opening with no supports that you position with X, Y, and Z coordinates. The default position is the center of the plate.You define the length and the width.

0S Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

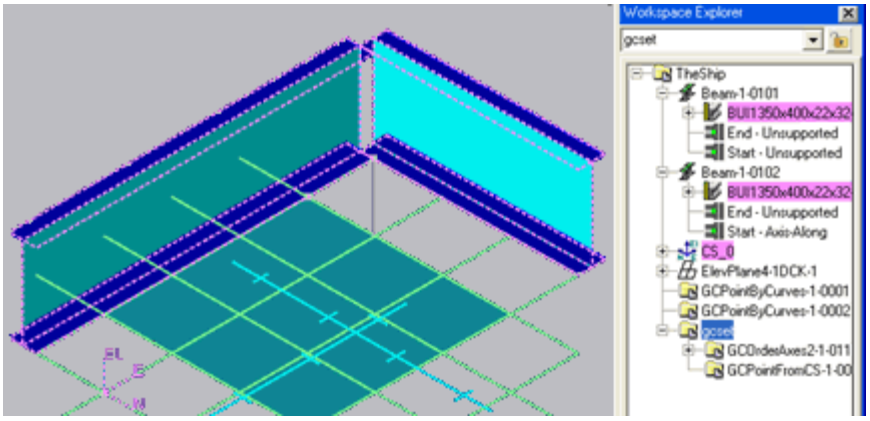
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length (or height) of the oval opening.

Width - Defines the width of the oval opening.

What do you want to do?

- *Place an oval opening with no supports* (on page 579)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

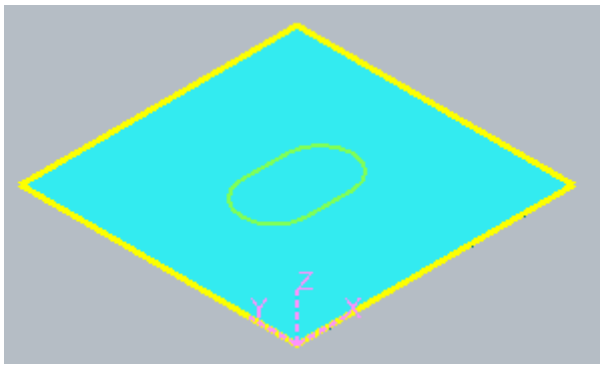
Place an oval opening with no supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **0S Oval** tab, click **0SOval HxW**.

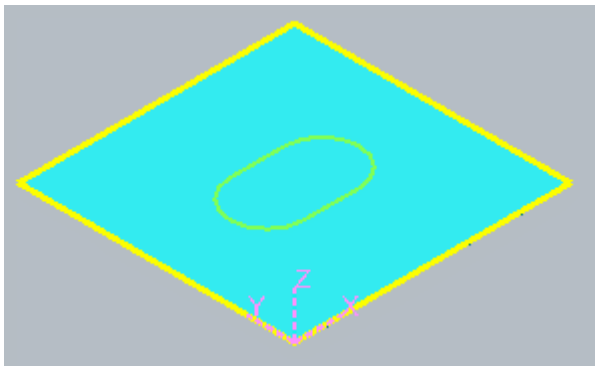
   The ribbon for the macro displays.
5. Select a coordinate system.

*A local coordinate system at the plate origin displays. A preview of the opening using the default length and width displays at the center of the plate. **X**, **Y**, and **Z** display the local coordinates at the center of the plate.*



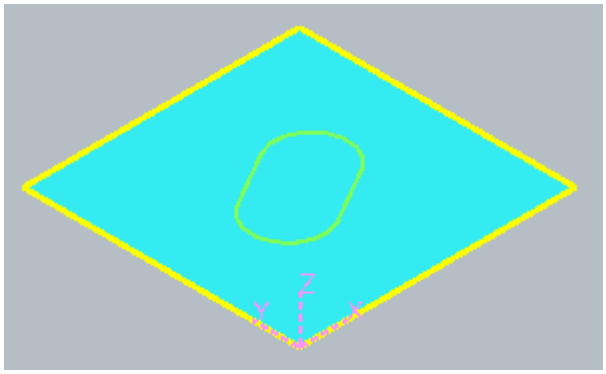
6. If required, type new values for **Length** and **Width**.

   The preview reflects the new opening size.



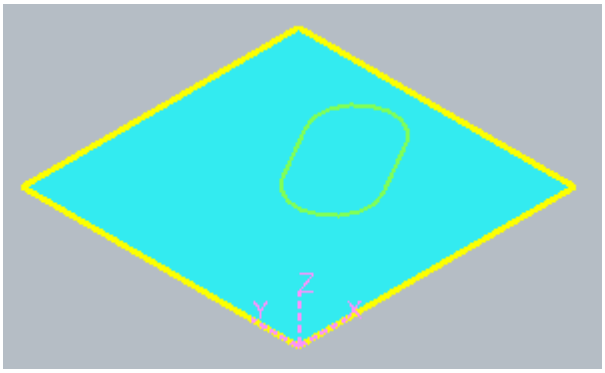
7. If required, type a new value for **Angle**.

   The preview reflects the new parameters.



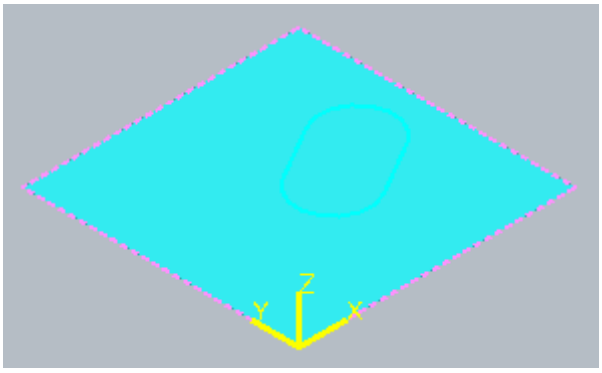
8. If required, type new **X**, **Y**, and **Z** values.

The preview reflects the new opening location.



9. Click **Continue**.

   *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

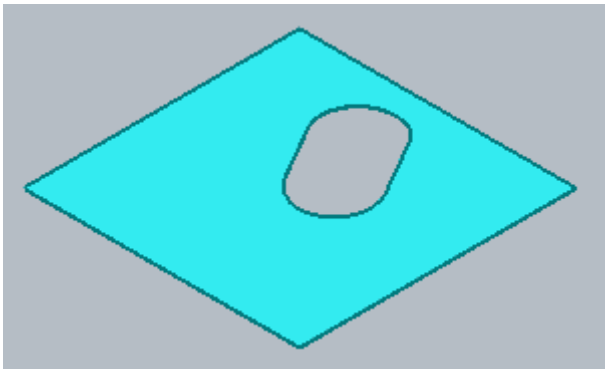
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

10. Click **Close**.

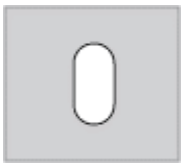
    *The ribbon display returns to the main **Openings** ribbon.*

11. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
12. Click **Finish**.

The software creates the opening.



Oval Opening with No Supports, a 500 mm Length, and a 250 mm Width



Tooltip: 0SOval 500x250

Creates a 500 mm by 250 mm oval opening with no supports that you position with X, Y, and Z coordinates. The default position is the center of the plate.

0S Fixed Dimension Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

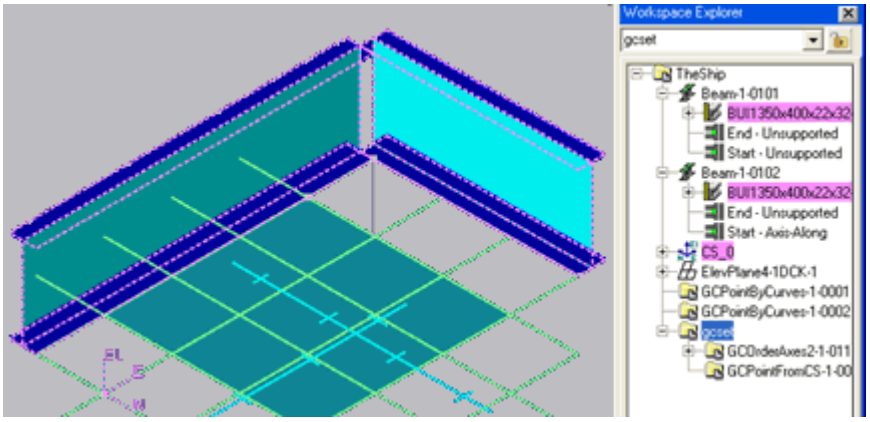
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

"The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of these options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length (or height) of the oval opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the oval opening.This option is unavailable and set to the fixed width of the macro.

What do you want to do?

- *Place an oval opening with no supports and a fixed height and width* (on page 584)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

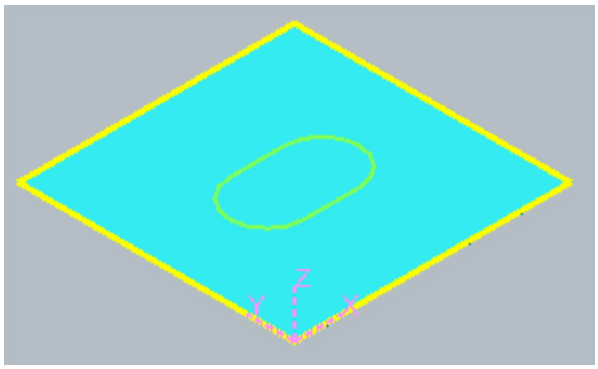
Place an oval opening with no supports and a fixed height and width

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **0SOval** tab, click **0SOval 500x250**, **0SOval 600x450**, or **0SOval 850x500**.

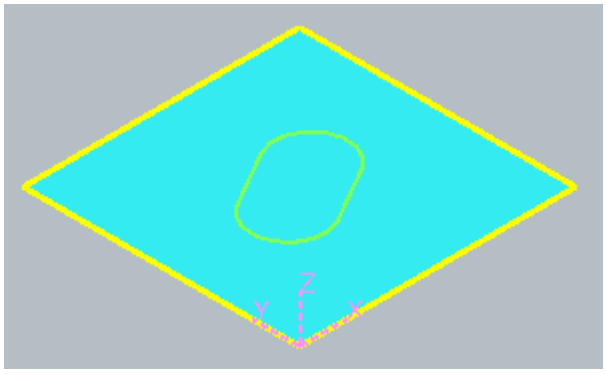
   The ribbon for the macro displays.
5. Select a coordinate system.

*A local coordinate system at the plate origin displays. A preview of the opening using the fixed length and width displays at the center of the plate. **X**, **Y**, and **Z** display the local coordinates at the center of the plate.*



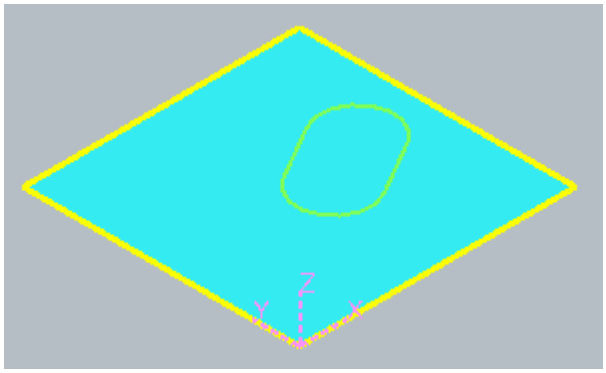
6. If required, type a new value for **Angle**.

   The preview reflects the new parameters.



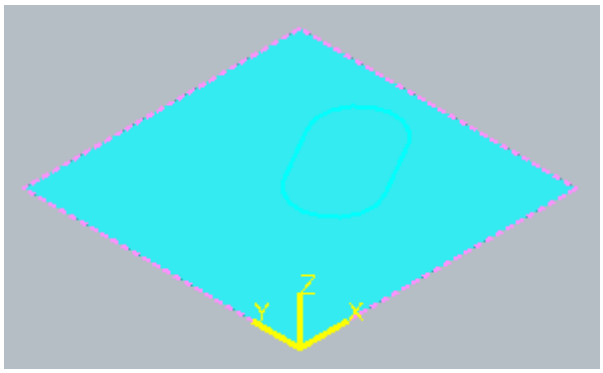
7. If required, type new **X**, **Y**, and **Z** values.

   The preview reflects the new opening location.



8. Click **Continue**.

*The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

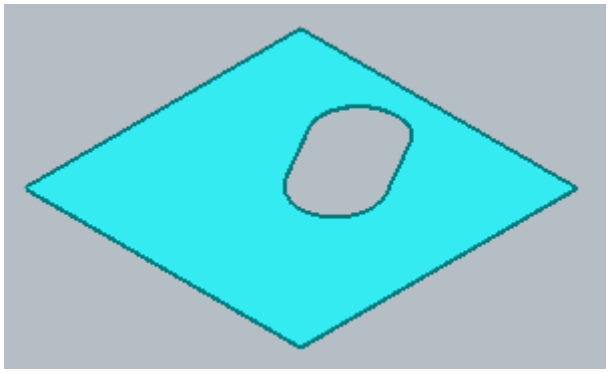
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

9. Click **Close**.

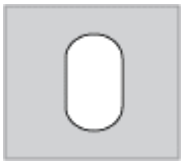
   *The ribbon display returns to the main **Openings** ribbon.*

10. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
11. Click **Finish**.

    The software creates the opening.



Oval Opening with No Supports, an 850 mm Length, and a 500 mm Width



Tooltip: 0SOval 850x500

Creates an 850 mm by 500 mm oval opening with no supports that you position with X, Y, and Z coordinates. The default position is the center of the plate.

0S Fixed Dimension Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

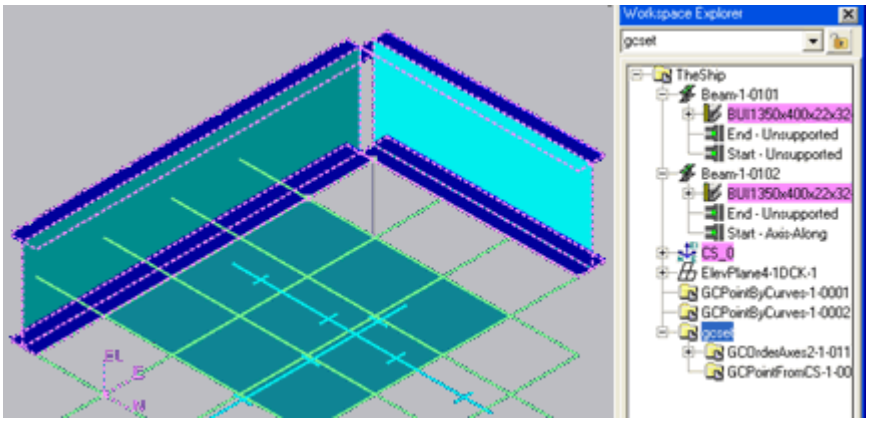
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of these options:

- **Normal** - Normal to the surface of the plate system.

- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length (or height) of the oval opening. This option is unavailable and set to the fixed length of the macro.

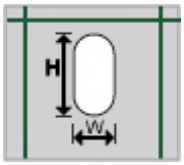
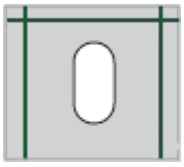
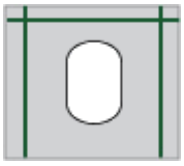
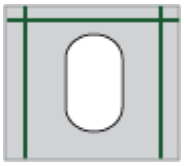
Width - Defines the width of the oval opening.This option is unavailable and set to the fixed width of the macro.

What do you want to do?

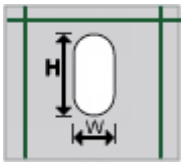
- *Place an oval opening with no supports and a fixed height and width* (on page 584)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

3S Oval Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place oval openings with three supports. The software positions a three-support opening by dimensions from the first support and is centers the opening between the second and third supports.

	**3S Oval HxW** - Creates an oval opening with three supports. You can define length and width. For more information, see *Oval Opening with Three Supports* (on page 590).
	**3S Oval 500x250** - Creates an oval opening with three supports. The length is fixed at 500 mm and the width is fixed at 250 mm. For more information, see *Oval Opening with Three Supports, a 500 mm Length, and a 250 mm Width* (on page 598).
	**3S Oval 600x450** - Creates an oval opening with three supports. The length is fixed at 600 mm and the width is fixed at 450 mm. For more information, see *Oval Opening with Three Supports, a 600 mm Length, and a 450 mm Width* (on page 605).
	**3S Oval 850x500** - Creates an oval opening with three supports. The length is fixed at 850 mm and the width is fixed at 500 mm. For more information, see *Oval Opening with Three Supports, an 850 mm Length, and a 500 mm Width* (on page 610).

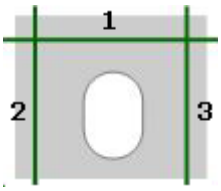
Oval Opening with Three Supports



Tooltip: Oval HxWxD

Creates an oval opening with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

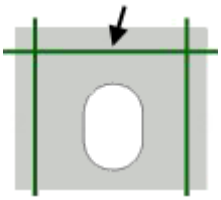
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

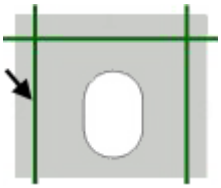
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

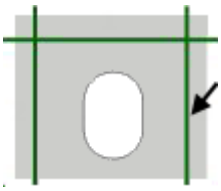
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



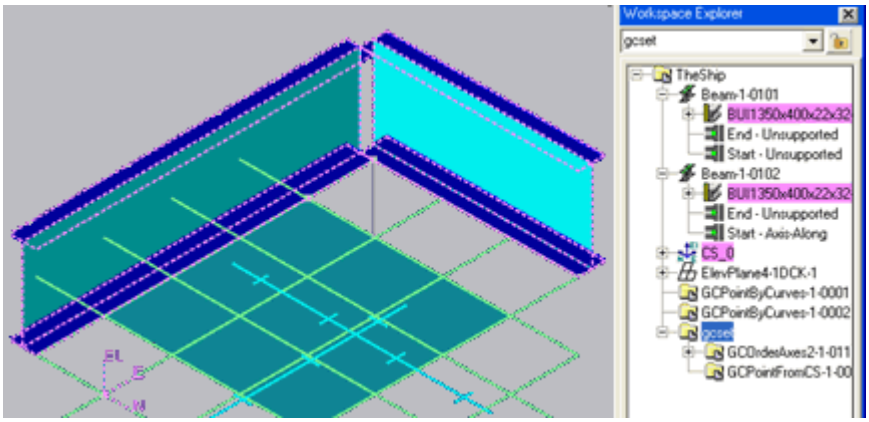
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

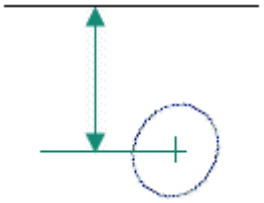
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

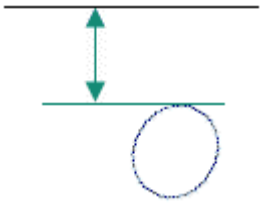
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



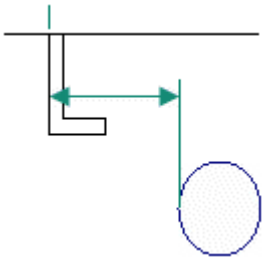
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



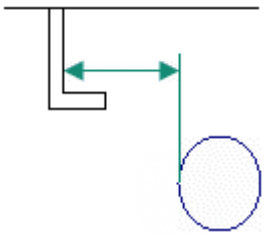
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

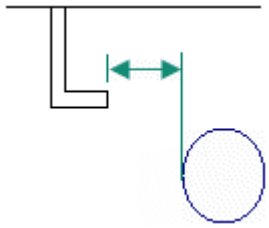
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

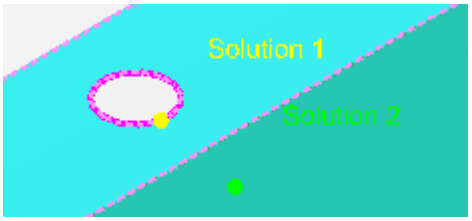
Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length (or height) of the oval opening.

Width - Defines the width of the oval opening.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place an oval opening with three supports* (on page 595)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

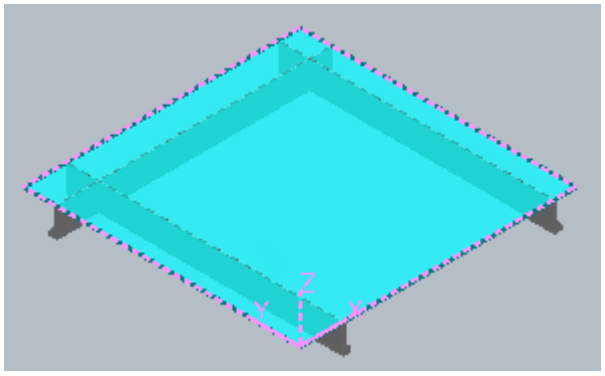
Place an oval opening with three supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

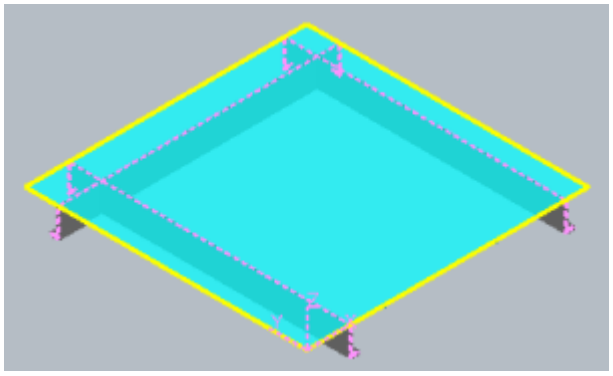
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **3S Oval** tab, click **Oval HxWxD**.

   The ribbon for the macro displays.
5. Select a coordinate system.

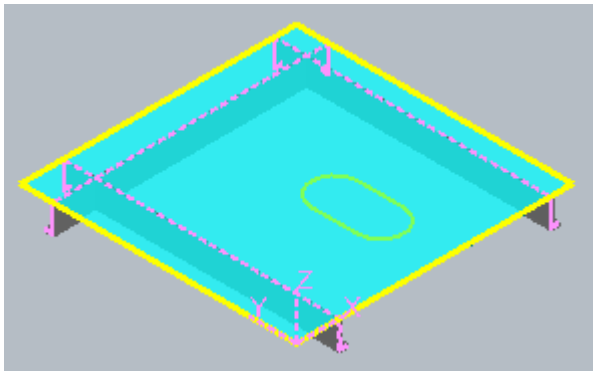
   A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the middle support, and then select the two side supports.



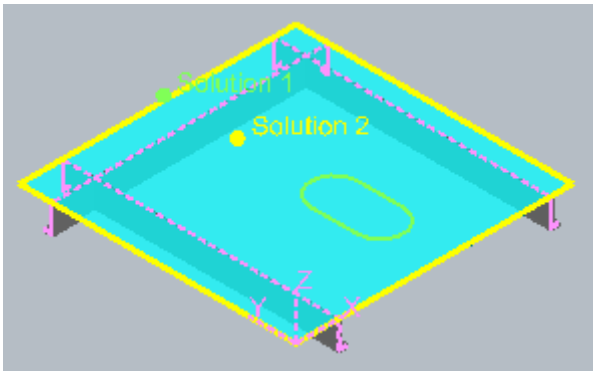
*If the default solution for **Distance Dir** defines geometry on the plate, a preview displays.*



NOTE *If the default solution for **Distance Dir** does not define geometry on the plate, then a message displays: "Make sure ProjDirection and the DistanceDir are valid and try again."*

8. If a solution is not possible or is not correct:
   a. Click **Distance Dir**.

      The possible solutions display.

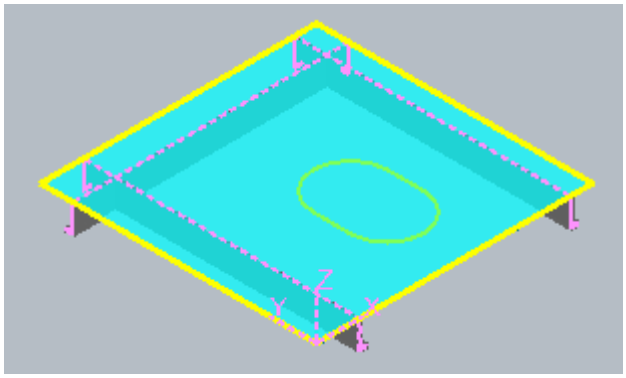


   b. Select the required solution for **Distance Dir**.

      The opening preview displays.

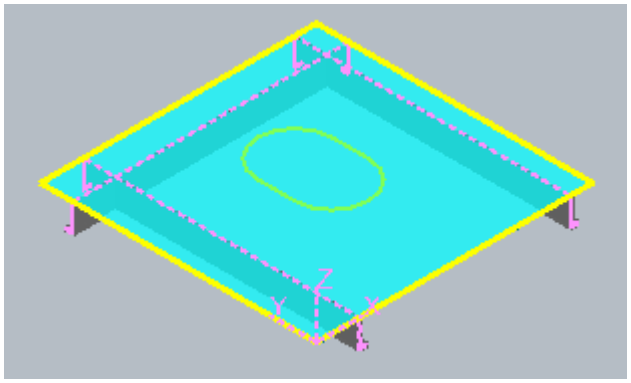
9. If required, type new values for the **Length** and **Width**.

   The preview reflects the opening size.



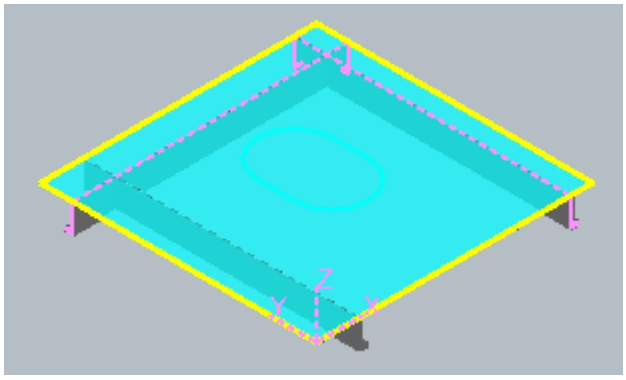
10. If required, change **Distance From** and **Distance** values to change the opening distance with respect to the middle support.

The preview reflects the new opening location.



11. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

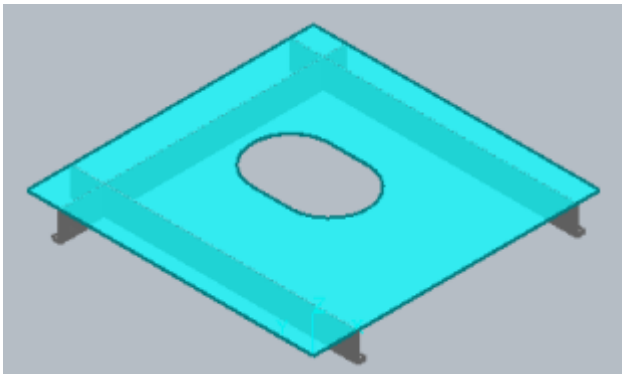
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

12. Click **Close**.

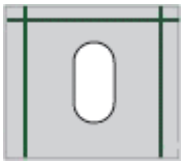
    *The ribbon display returns to the main **Openings** ribbon.*

13. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
14. Click **Finish**.

The software creates the opening.



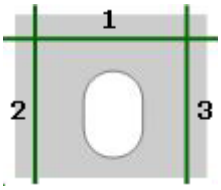
Oval Opening with Three Supports, a 500 mm Length, and a 250 mm Width



Tooltip: Oval 500x250

Creates a 500 mm by 250 mm oval opening with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Dimension Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

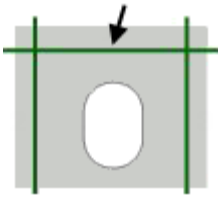
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

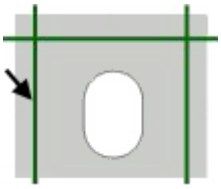
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

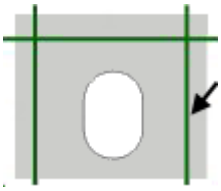
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



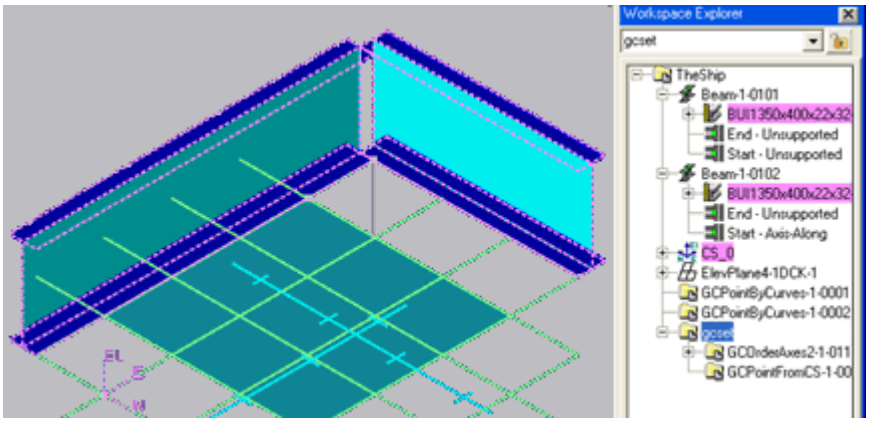
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

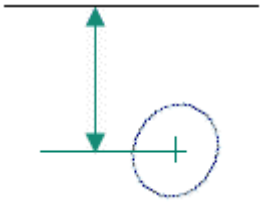
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

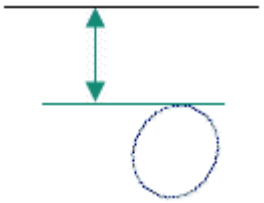
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



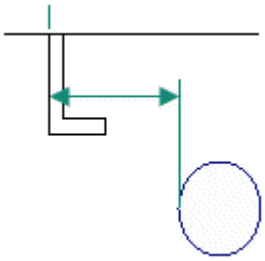
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



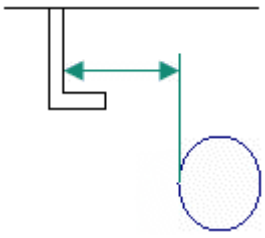
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

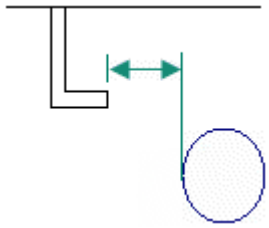
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

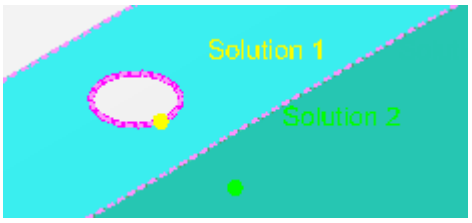
Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length (or height) of the oval opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the oval opening. This option is unavailable and set to the fixed width of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place an oval opening with three supports and a fixed height and width* (on page 603)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

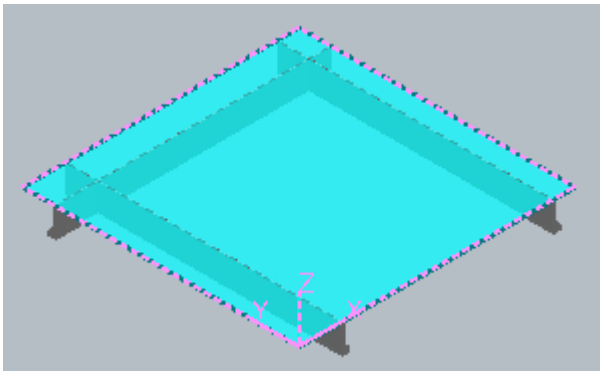
Place an oval opening with three supports and a fixed height and width

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

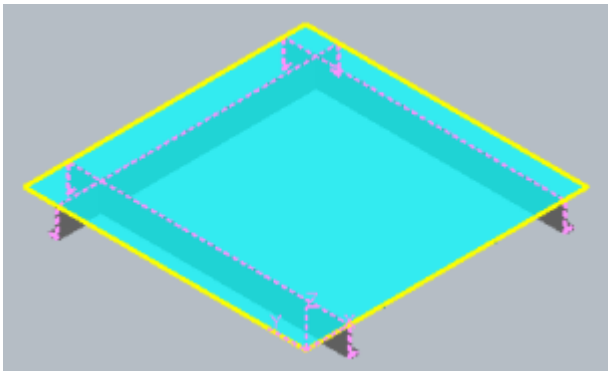
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **3S Oval** tab, click **Oval 500x250**, **Oval 600x450**, or **Oval 850x500**.

   The ribbon for the macro displays.
5. Select a coordinate system.

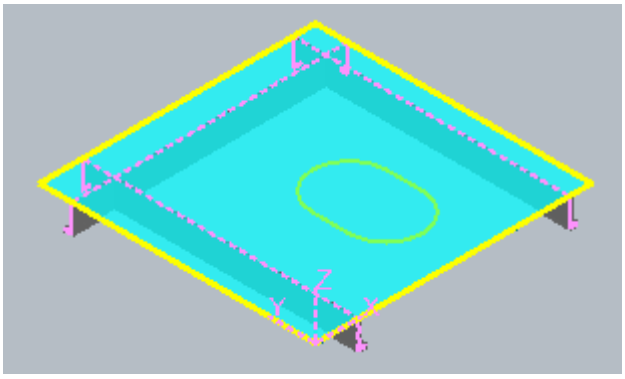
   A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the middle support, and then select the two side supports.



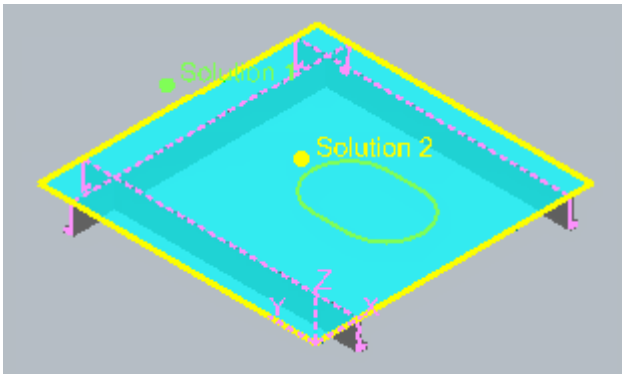
*If the default solution for **Distance Dir** defines geometry on the plate, a preview displays.*



NOTE *If the default solution for **Distance Dir** does not define geometry on the plate, then a message displays: "Make sure ProjDirection and the DistanceDir are valid and try again."*

8. If a solution is not possible or is not correct:
   a. Click **Distance Dir**.

      The possible solutions display.

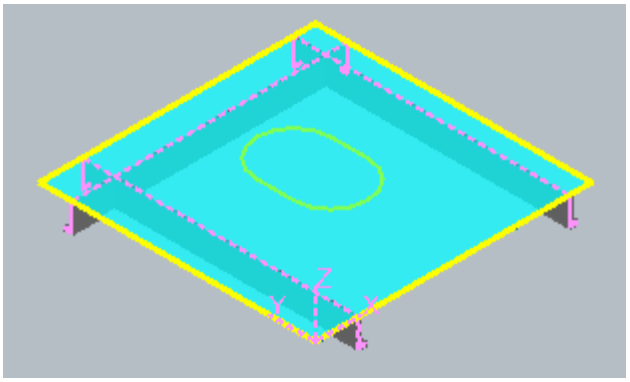


   b. Select the required solution for **Distance Dir**.

      The opening preview displays.

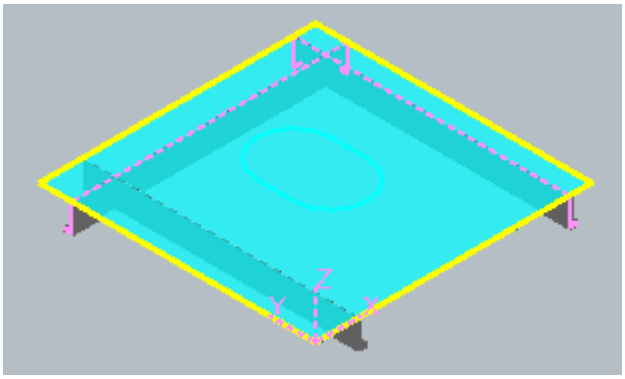
9. If required, change **Distance From** and **Distance** values to change the opening distance with respect to the middle support.

   The preview reflects the new opening location.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

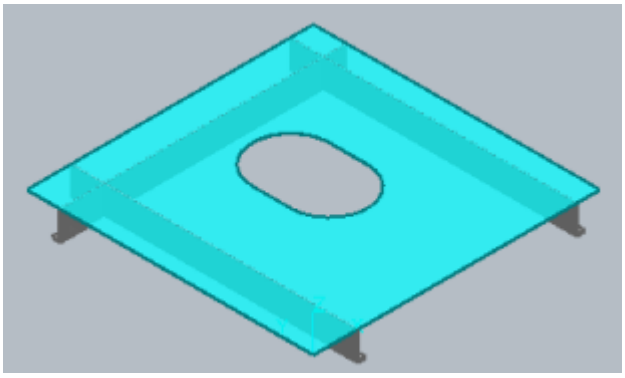
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

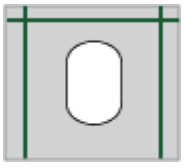
    *The ribbon display returns to the main **Openings** ribbon.*

12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

    The software creates the opening.



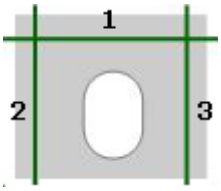
Oval Opening with Three Supports, a 600 mm Length, and a 450 mm Width



Tooltip: Oval 600x450

Creates a 600 mm by 450 mm oval opening with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Dimension Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

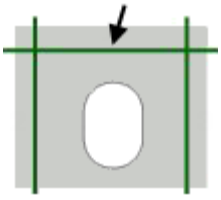
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

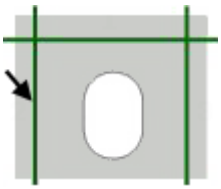
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

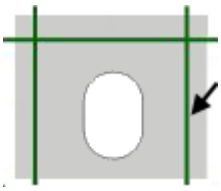
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



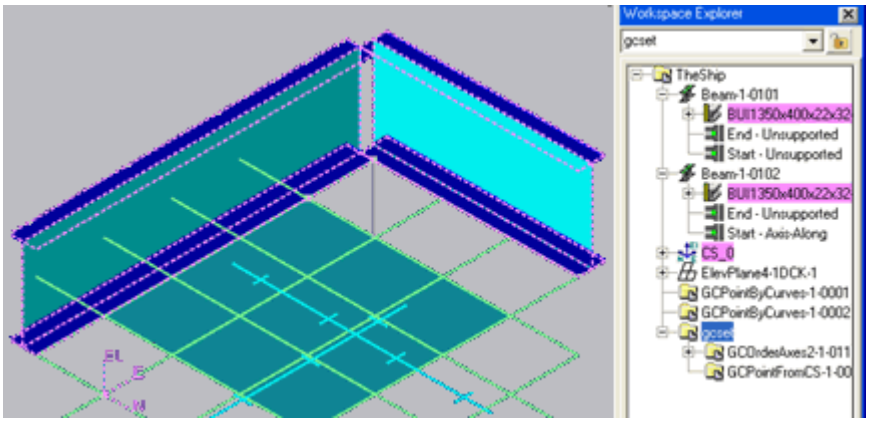
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

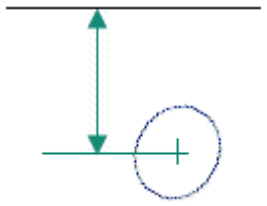
- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

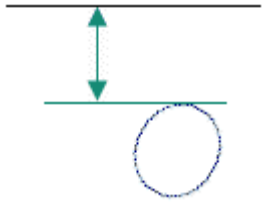
  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



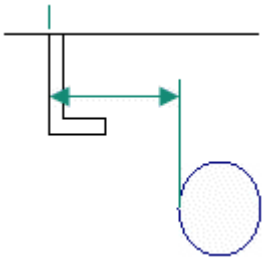
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



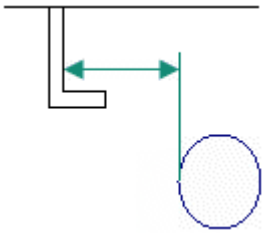
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

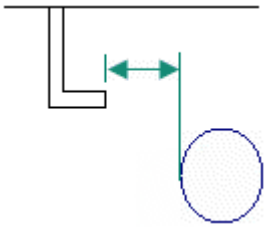
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

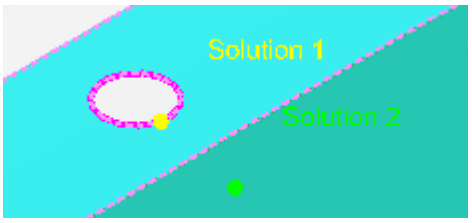
Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length (or height) of the oval opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the oval opening. This option is unavailable and set to the fixed width of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

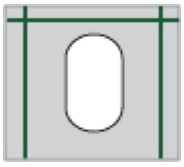
Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place an oval opening with three supports and a fixed height and width* (on page 603)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

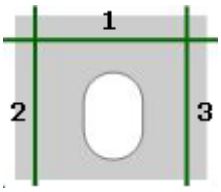
Oval Opening with Three Supports, an 850 mm Length, and a 500 mm Width



Tooltip: Oval 850x500

Creates an 850 mm by 500 mm oval opening with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Dimension Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

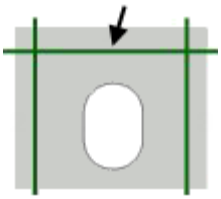
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

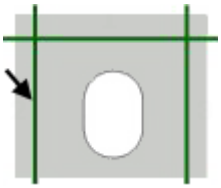
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

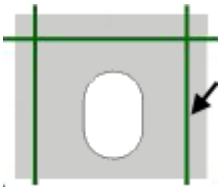
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



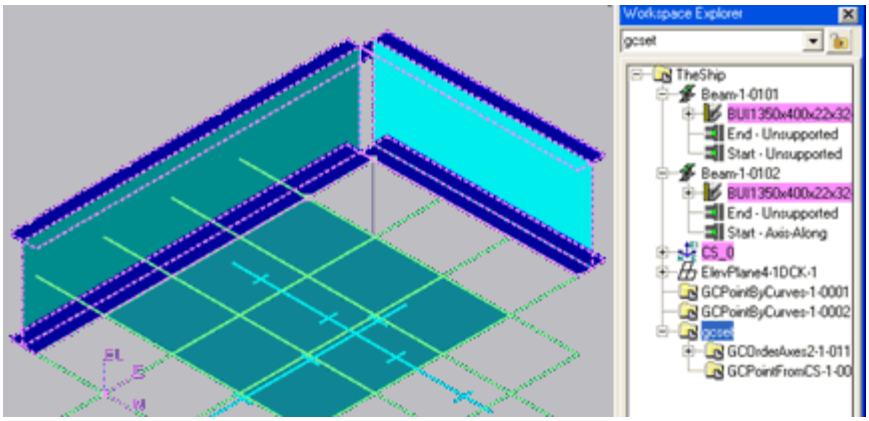
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

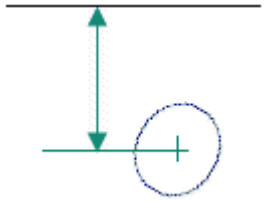
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

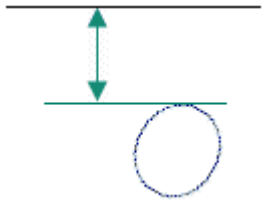
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



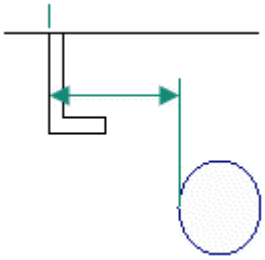
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



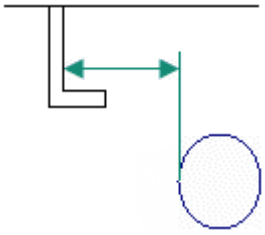
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

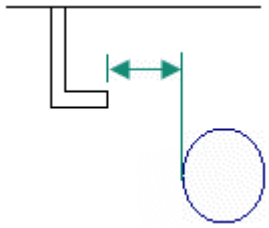
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

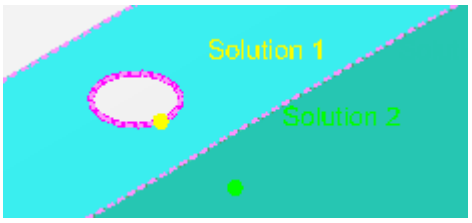
Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length (or height) of the oval opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the oval opening. This option is unavailable and set to the fixed width of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.

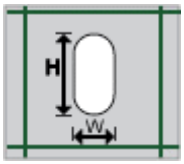
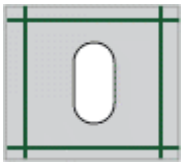
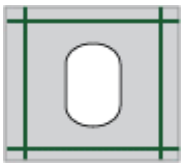
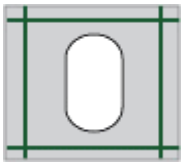


What do you want to do?

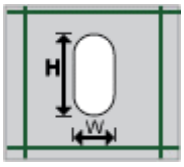
- *Place an oval opening with three supports and a fixed height and width* (on page 603)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

4S Oval Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place oval openings with four supports. The software centers a four-support opening between opposing supports.

	**4S Oval HxW** - Creates an oval opening with four supports. You can define length and width. For more information, see *Oval Opening with Four Supports* (on page 615).
	**4S Oval 500x250** - Creates an oval opening with four supports. The length is fixed at 500 mm and the width is fixed at 250 mm. For more information, see *Oval Opening with Four Supports, a 500 mm Length, and a 250 mm Width* (on page 622).
	**4S Oval 600x450** - Creates an oval opening with four supports. The length is fixed at 600 mm and the width is fixed at 450 mm. For more information, see *Oval Opening with Four Supports, a 600 mm Length, and a 450 mm Width* (on page 629).
	**4S Oval 850x500** - Creates an oval opening with four supports. The length is fixed at 850 mm and the width is fixed at 500 mm. For more information, see *Oval Opening with Four Supports, an 850 mm Length, and a 500 mm Width* (on page 633).

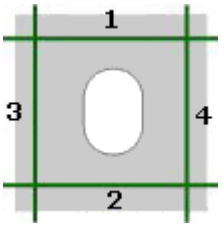
Oval Opening with Four Supports



Tooltip: 4SOval HxW

Creates an oval opening with four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

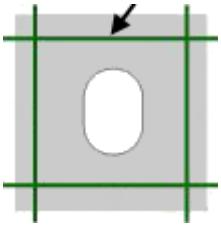
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

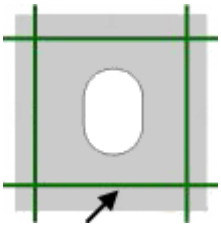
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

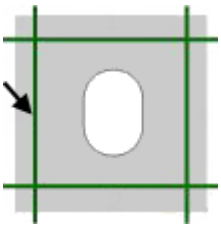
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



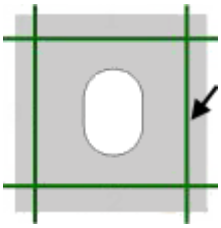
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



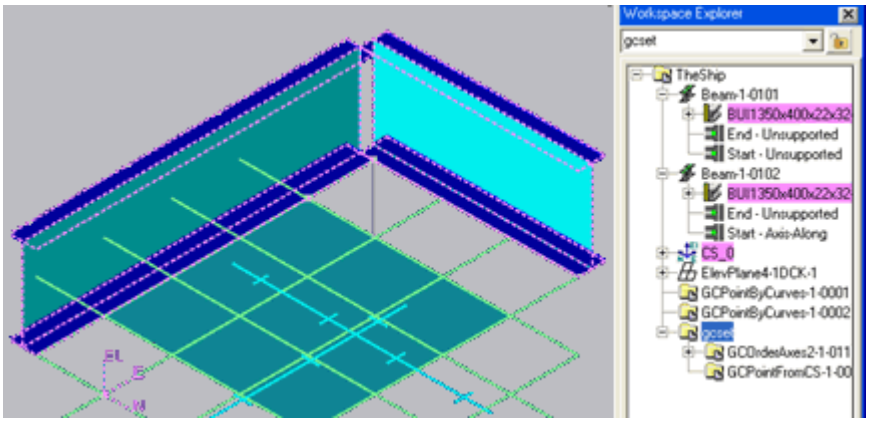
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

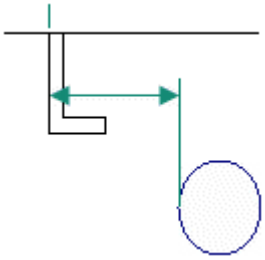
- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

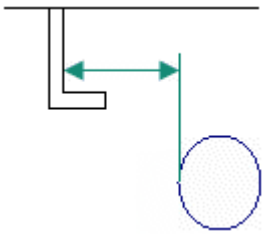
  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

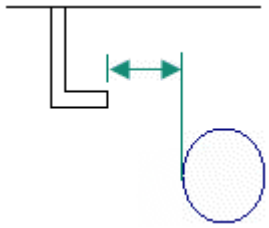
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length of the oval opening.

Width - Defines the width of the oval opening.

What do you want to do?

- *Place an oval opening with four supports* (on page 619)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

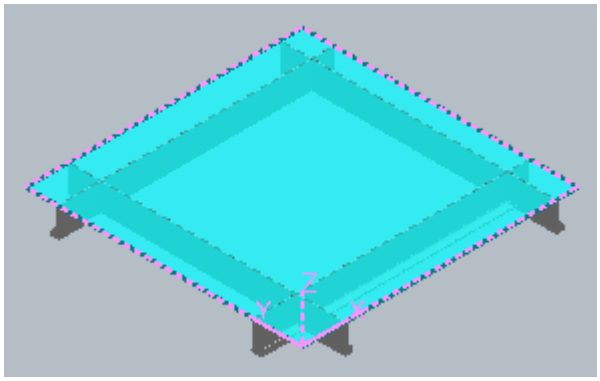
Place an oval opening with four supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

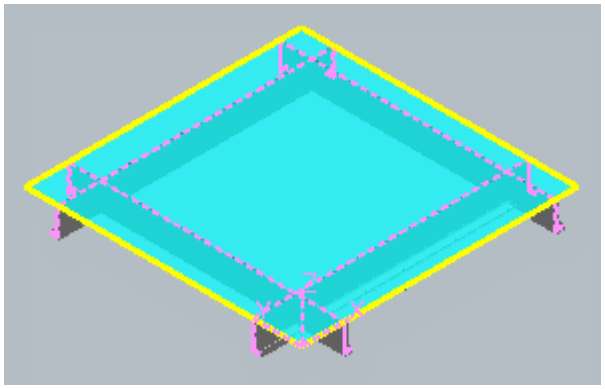
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **4S Oval** tab, click **4SOval HxW**.

   The ribbon for the macro displays.
5. Select a coordinate system.

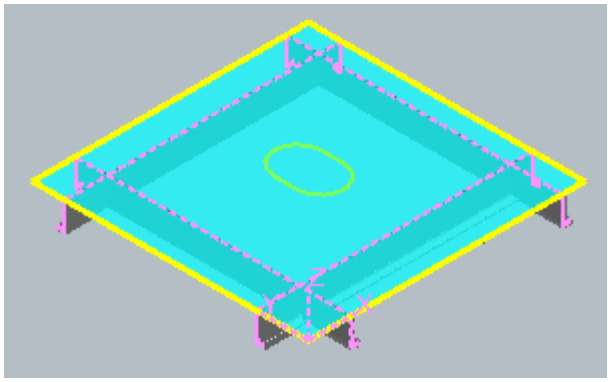
A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the first support, a second support opposite the first support, a third support, and a fourth support opposite the third support.

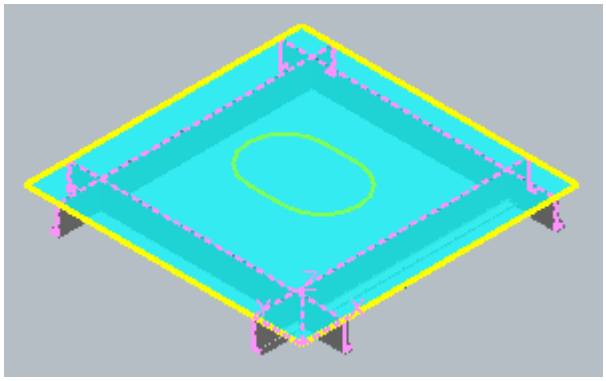


The opening preview displays.



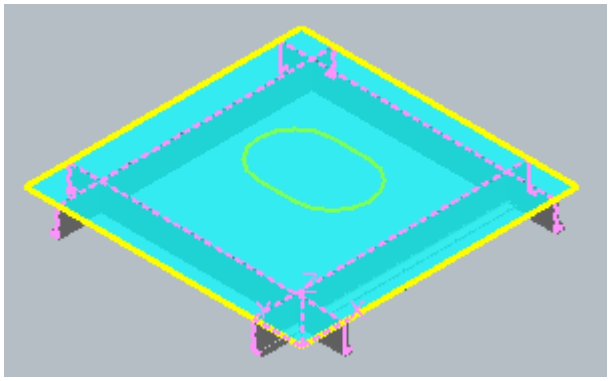
8. If required, type new values for **Angle**, **Length**, and **Width**.

The preview reflects the new parameters.



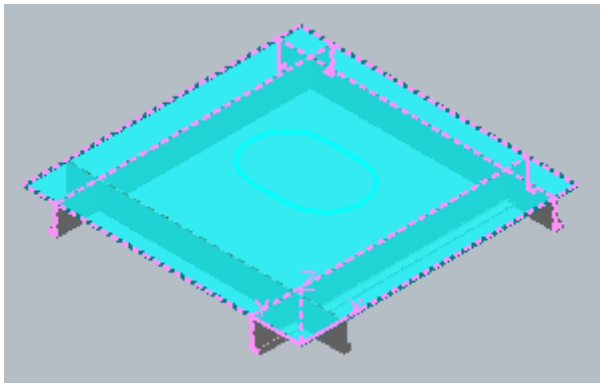
9. If required, change the **Dimension Type** value to change the opening distance from the supports.

   The preview reflects the new opening location.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

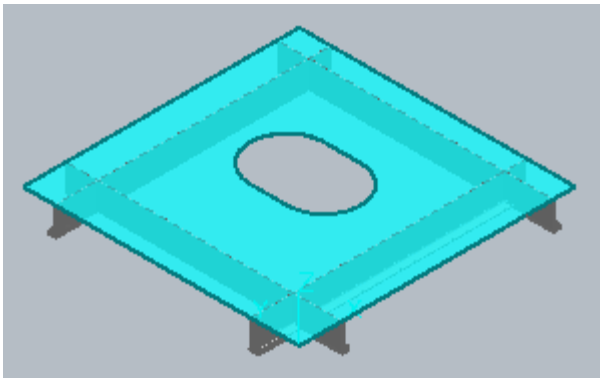
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

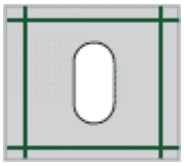
*The ribbon display returns to the main **Openings** ribbon.*

12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

    The software creates the opening.



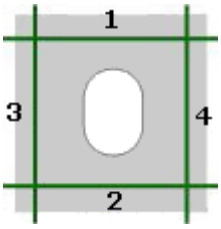
Oval Opening with Four Supports, a 500 mm Length, and a 250 mm Width



Tooltip: 4SOval 500x250

Creates a 500 mm by 250 mm oval opening with four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Dimension Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

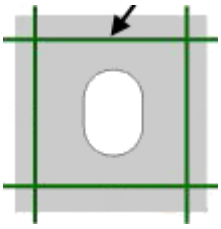
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

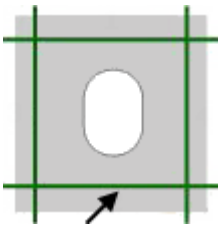
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

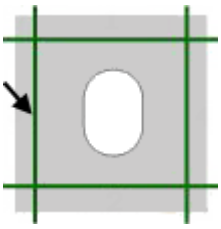
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



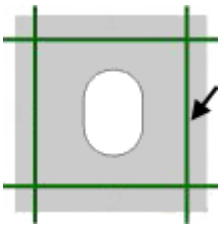
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



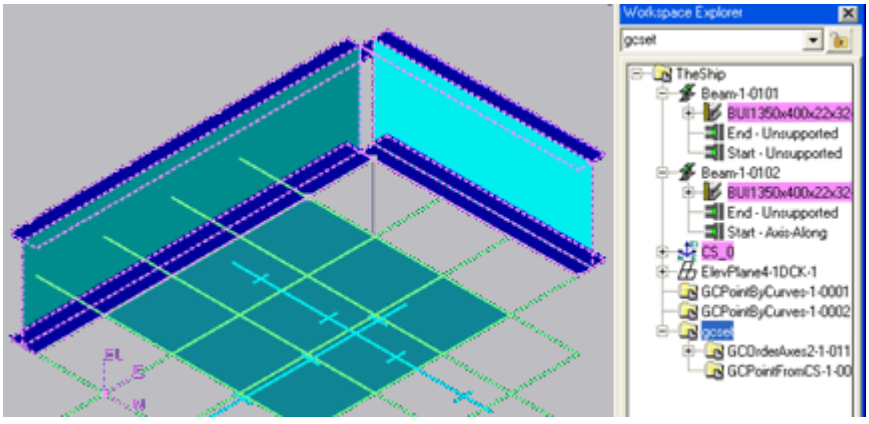
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

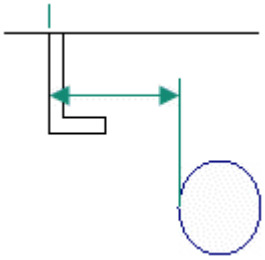
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

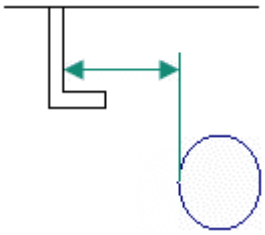
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

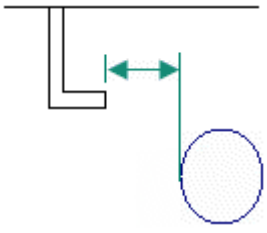
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length of the oval opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the oval opening. This option is unavailable and set to the fixed width of the macro.

What do you want to do?

- *Place an oval opening with four supports and a fixed height and width* (on page 626)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

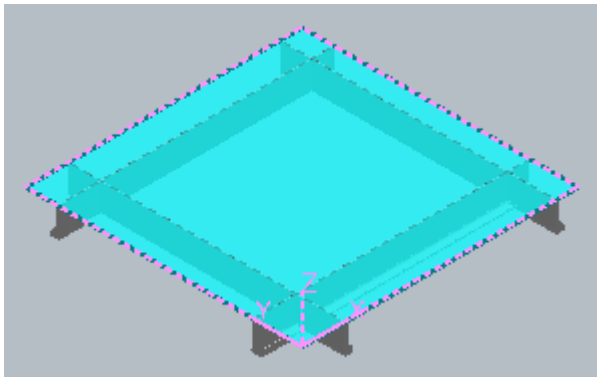
Place an oval opening with four supports and a fixed height and width

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

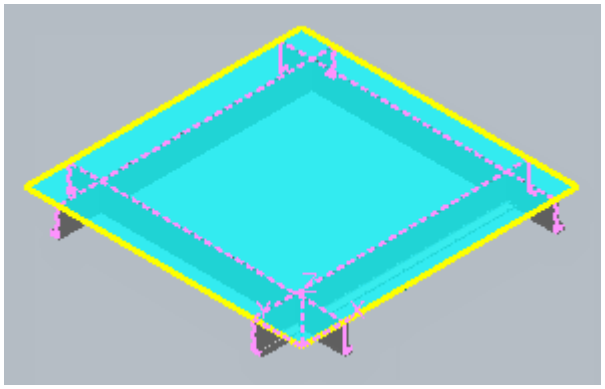
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **4S Oval** tab, click **4SOval 500x250**, **4SOval 600x450**, or **4SOval 850x500**.

   The ribbon for the macro displays.
5. Select a coordinate system.

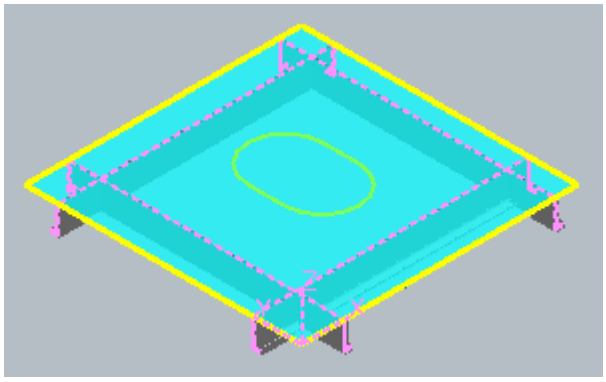
A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the first support, a second support opposite the first support, a third support, and a fourth support opposite the third support.

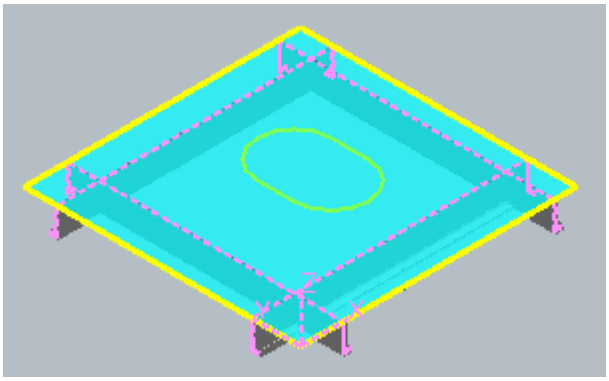


The opening preview displays.



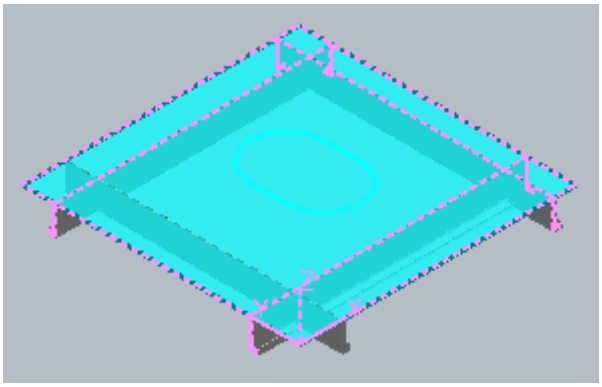
8. If required, type a new value for **Angle**.
9. If required, change the **Dimension Type** value to change the opening distance from the supports.

The preview reflects the new opening location.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

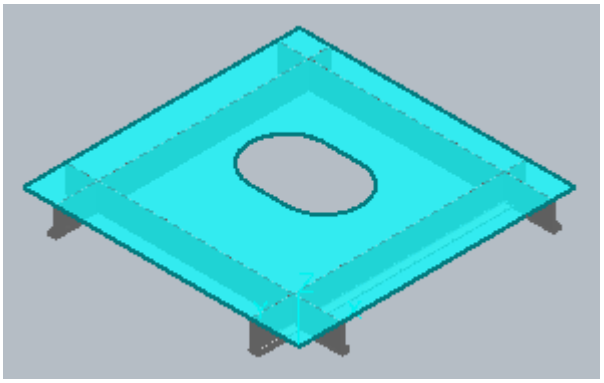
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

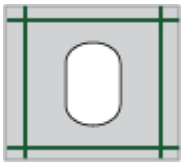
    *The ribbon display returns to the main **Openings** ribbon.*

12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

The software creates the opening.



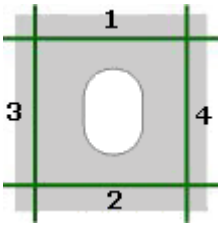
Oval Opening with Four Supports, a 600 mm Length, and a 450 mm Width



Tooltip: 4SOval 600x450

Creates a 600 mm by 450 mm oval opening with four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Dimension Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

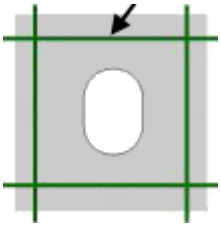
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

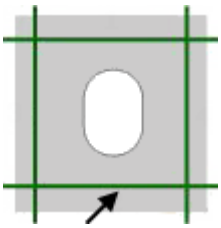
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

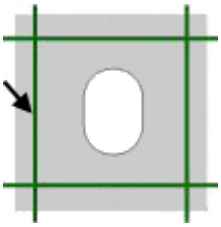
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



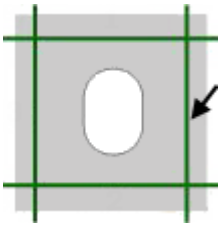
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



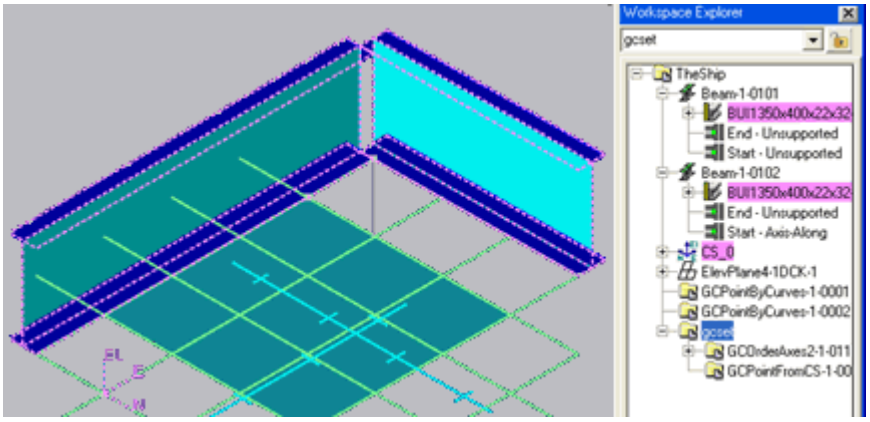
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

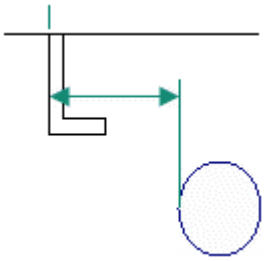
"The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and does not display when editing an object.

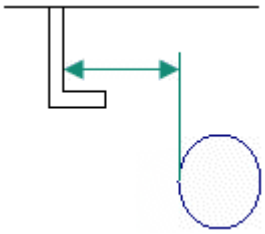
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

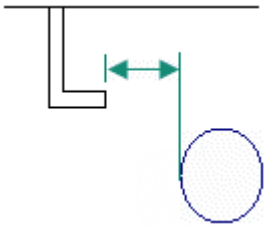
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

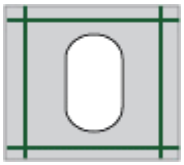
Length - Defines the length of the oval opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the oval opening. This option is unavailable and set to the fixed width of the macro.

What do you want to do?

- *Place an oval opening with four supports and a fixed height and width* (on page 626)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

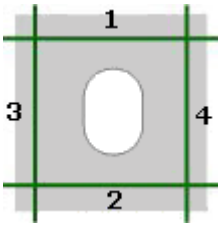
Oval Opening with Four Supports, an 850 mm Length, and a 500 mm Width



Tooltip: 4SOval 850x500

Creates an 850 mm by 500 mm oval opening with four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Dimension Oval Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

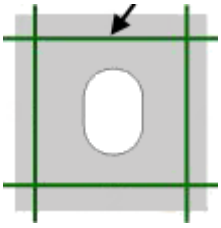
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

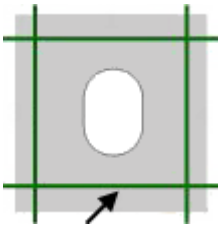
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

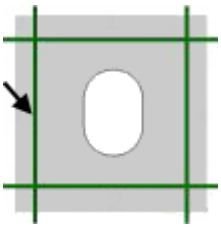
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



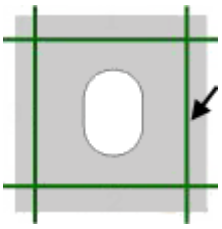
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



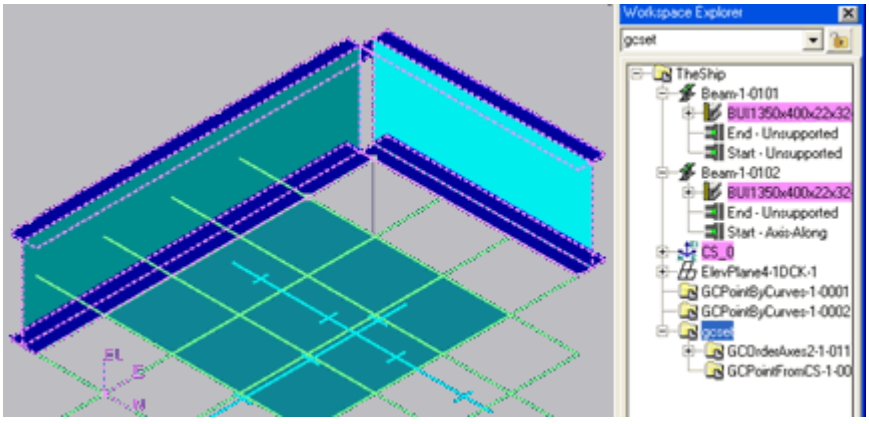
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

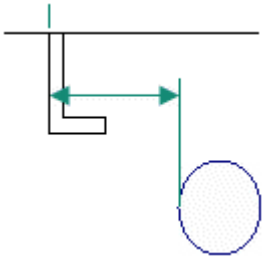
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

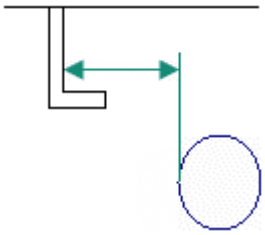
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

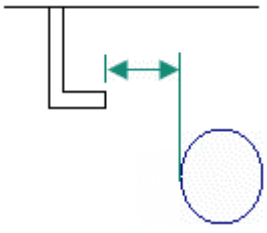
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Oval**.

Length - Defines the length of the oval opening. This option is unavailable and set to the fixed length of the macro.

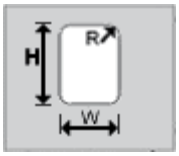
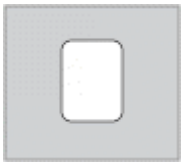
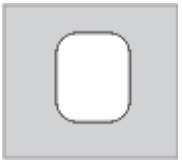
Width - Defines the width of the oval opening. This option is unavailable and set to the fixed width of the macro.

What do you want to do?

- *Place an oval opening with four supports and a fixed height and width* (on page 626)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

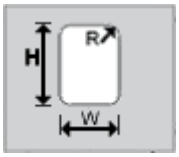
0S Rectangle Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place rectangular openings with no supports. The software positions a no-support opening by X-, Y-, and Z-coordinates. The default position is the center of the plate.

	**0S Rectangle HxWxR** - Creates a rounded rectangular opening with no supports. You can define the length, the width, and the corner radius. For more information, see *Rectangular Opening with No Supports* (on page 638).
	**0S Rectangle 400x300x100** - Creates a rounded rectangular opening with no supports. The length is fixed at 400 mm, the width is fixed at 300 mm, and the corner radius is fixed at 100 mm. For more information, see *Rectangular Opening with No Supports, a 400 mm Length, and a 300 mm Width* (on page 643).
	**0S Rectangle 600x500x200** - Creates a rounded rectangular opening with no supports. The length is fixed at 600 mm, the width is fixed at 500 mm, and the corner radius is fixed at 200 mm. For more information, see *Rectangular Opening with No Supports, a 600 mm Length, and a 500 mm Width* (on page 647).

	**0S Rectangle 1350x700x150** - Creates a rounded rectangular opening with no supports. The length is fixed at 1350 mm, the width is fixed at 700 mm , and the corner radius is fixed at 150 mm. For more information, see *Rectangular Opening with No Supports, a 1350 mm Length, and a 700 mm Width* (on page 650).

Rectangular Opening with No Supports



Tooltip: 0SRect HxWxR

Creates a rectangular opening with no supports that you position with X, Y, and Z coordinates. The default position is the center of the plate. You define the length, the width, and the corner radius.

0S Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

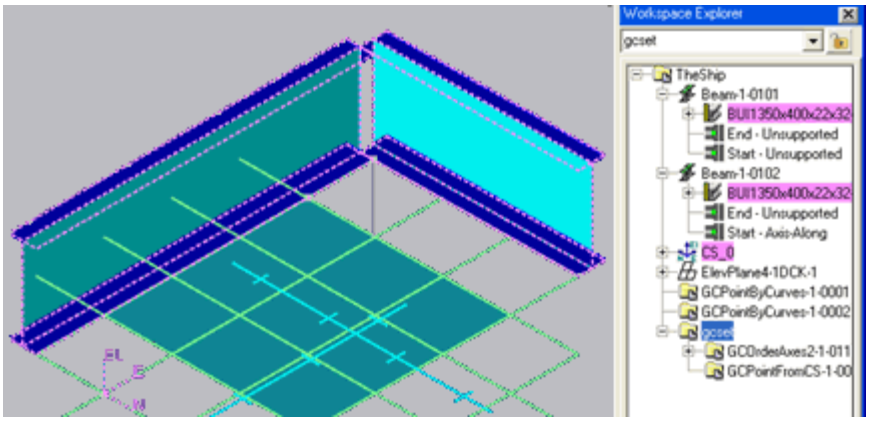
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

Length - Defines the length (or height) of the rounded rectangular opening.

Width - Defines the width of the rounded rectangular opening.

Rounded Rad - Defines the corner radius of the rounded rectangular opening.

What do you want to do?

- *Place a rectangular opening with no supports* (on page 640)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

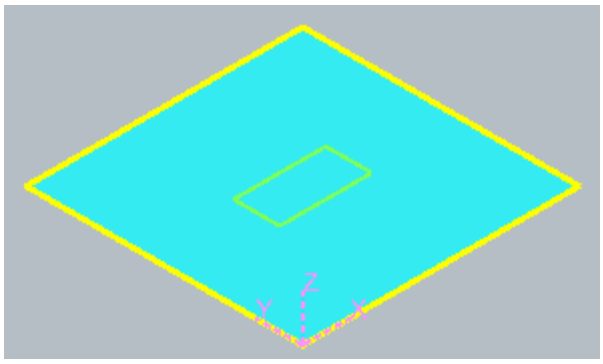
Place a rectangular opening with no supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **0S Rectangle** tab, click **0SRect HxWxR**.

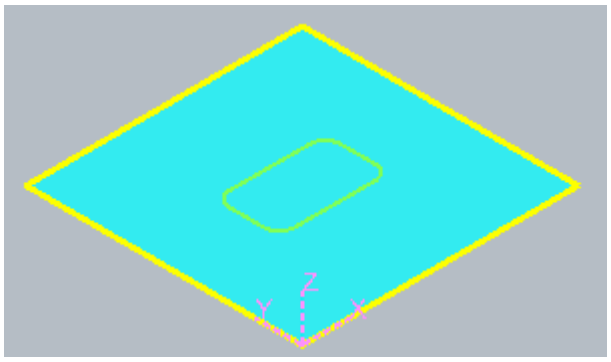
   The ribbon for the macro displays.
5. Select a coordinate system.

*A local coordinate system at the plate origin displays. A preview of the opening using the default size values displays at the center of the plate. **X**, **Y**, and **Z** display the local coordinates at the center of the plate.*



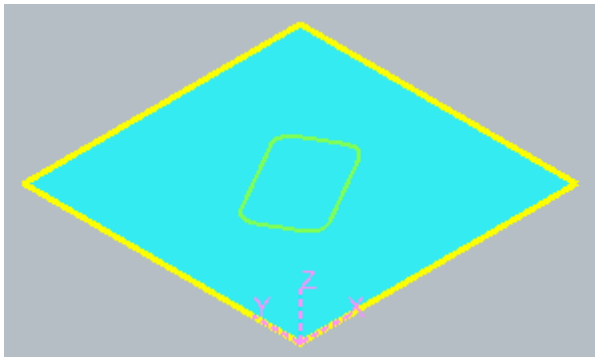
6. If required, type new values for **Length**, **Width**, and **Rounded Rad**.

   The preview reflects the new opening size.



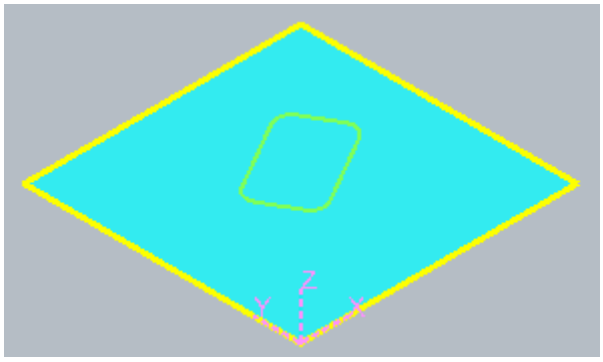
7. If required, type a new value for **Angle**.

   The preview reflects the new angle.



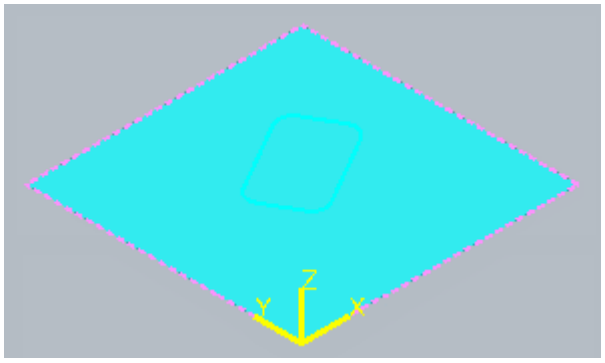
8. If required, type new **X**, **Y**, and **Z** values.

The preview reflects the new opening location.



9. Click **Continue**.

   *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

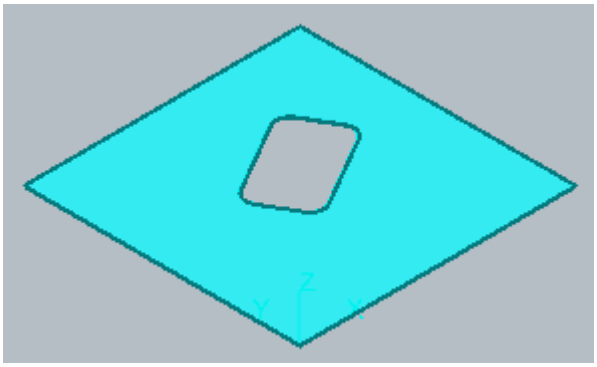
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

10. Click **Close**.

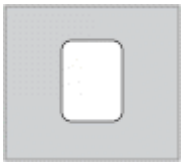
    *The ribbon display returns to the main **Openings** ribbon.*

11. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
12. Click **Finish**.

The software creates the opening.



Rectangular Opening with No Supports, a 400 mm Length, and a 300 mm Width



Tooltip: 0SRect 400x300x100

Creates a 400 mm by 300 mm rectangular opening with 100 mm radius corners and no supports that you position with X, Y, and Z coordinates. The default position is the center of the plate.

0S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

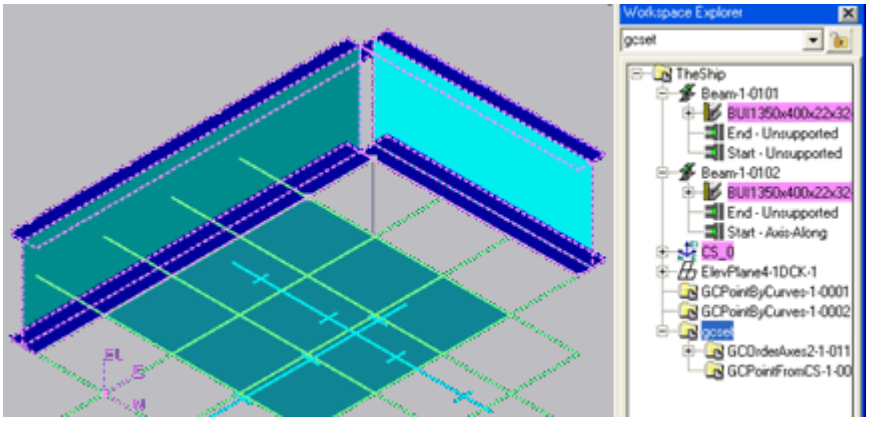
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

"The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

Length - Defines the length (or height) of the rounded rectangular opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the rounded rectangular opening. This option is unavailable and set to the fixed width of the macro.

Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

- *Place a rectangular opening with no supports and a fixed height and width* (on page 645)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

Place a rectangular opening with no supports and a fixed height and width

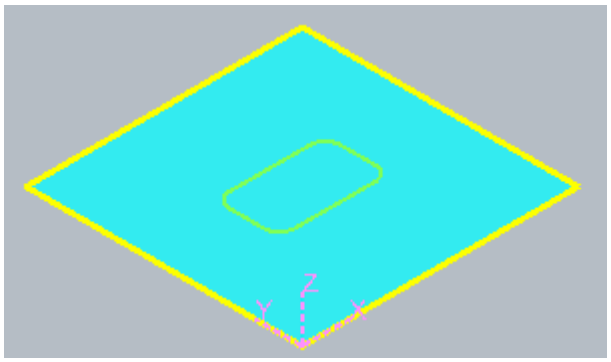
1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **0S Rectangle** tab, click **0SRect 400x300x100**, **0SRect 600x500x200**, or **0SRect 1350x700x150**.

The ribbon for the macro displays.

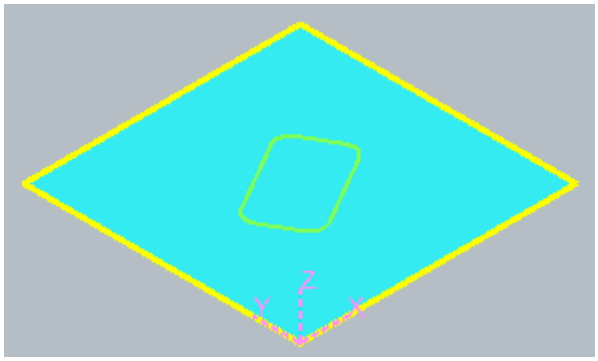
5. Select a coordinate system.

   *A local coordinate system at the plate origin displays. A preview of the opening using the fixed size values displays at the center of the plate. **X**, **Y**, and **Z** display the local coordinates at the center of the plate.*



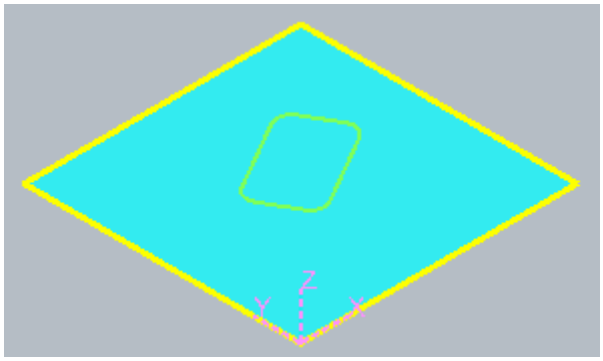
6. If required, type a new value for **Angle**.

   The preview reflects the new angle.



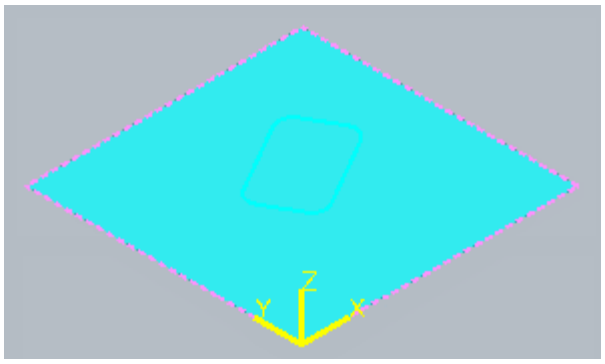
7. If required, type new **X**, **Y**, and **Z** values.

   The preview reflects the new opening location.



8. Click **Continue**.

*The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

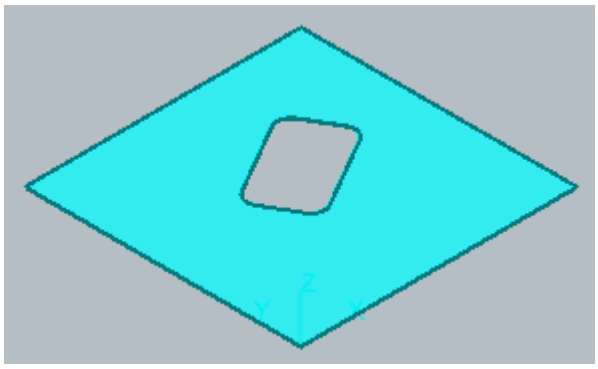
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

9. Click **Close**.

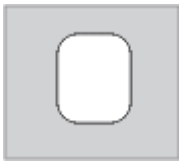
   *The ribbon display returns to the main **Openings** ribbon.*

10. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
11. Click **Finish**.

    The software creates the opening.



Rectangular Opening with No Supports, a 600 mm Length, and a 500 mm Width



Tooltip: 0SRect 600x500x200

Creates a 600 mm by 500 mm rectangular opening with 200 mm radius corners, with no supports, and positioned by X, Y, and Z coordinates. The default position is the center of the plate.

0S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

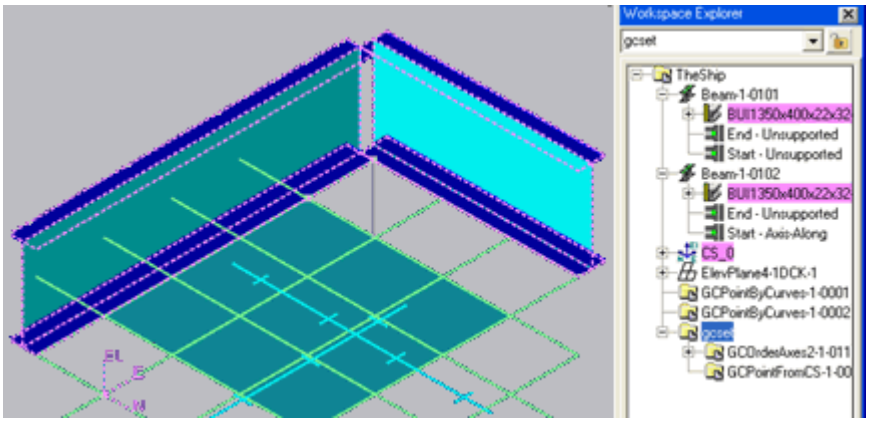
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.

- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

Length - Defines the length (or height) of the rounded rectangular opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the rounded rectangular opening. This option is unavailable and set to the fixed width of the macro.

Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

- *Place a rectangular opening with no supports and a fixed height and width* (on page 645)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

Rectangular Opening with No Supports, a 1350 mm Length, and a 700 mm Width



Tooltip: 0SRect 1350x700x150

Creates a 1350 mm by 700 mm rectangular opening with 150 mm radius corners and no supports that you position with X, Y, and Z coordinates. The default position is the center of the plate.

0S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

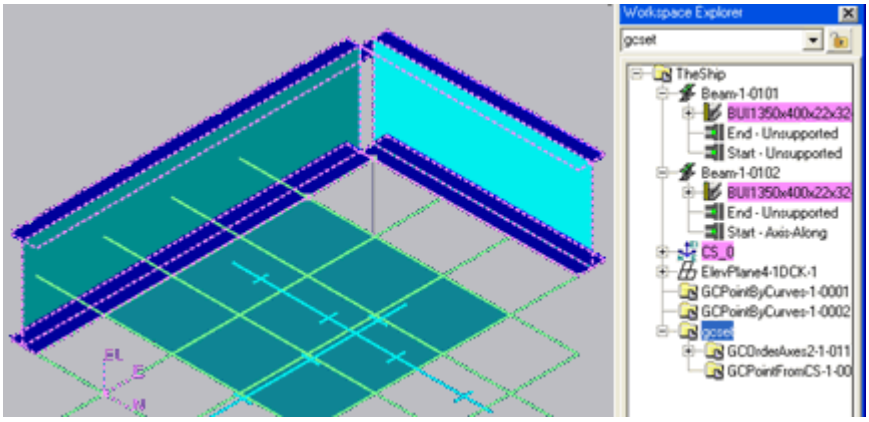
2 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

X - Defines the X-coordinate of the selected coordinate system for the opening center.

Y - Defines the Y-coordinate of the selected coordinate system for the opening center.

Z - Defines the Z-coordinate of the selected coordinate system for the opening center.

Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about its center. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

Length - Defines the length (or height) of the rounded rectangular opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the rounded rectangular opening. This option is unavailable and set to the fixed width of the macro.

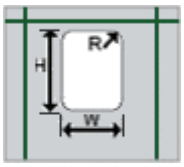
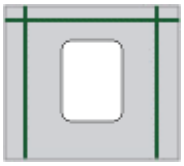
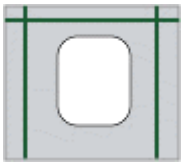
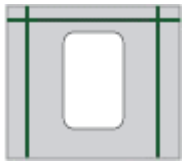
Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

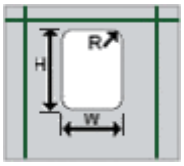
- *Place a rectangular opening with no supports and a fixed height and width* (on page 645)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

3S Rectangle Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place rectangular openings with three supports. The software positions a three-support opening by dimensions from the first support and is centers the opening between the second and third supports.

	**3S Rectangle HxWxR** - Creates a rounded rectangular opening with three supports. You can define the length, the width, and the corner radius. For more information, see *Rectangular Opening with Three Supports* (on page 653).
	**3S Rectangle 400x300x100** - Creates a rounded rectangular opening with three supports. The length is fixed at 400 mm, the width is fixed at 300 mm, and the corner radius is fixed at 100 mm. For more information, see *Rectangular Opening with Three Supports, a 400 mm Length, and a 300 mm Width* (on page 662).
	**3S Rectangle 600x500x200** - Creates a rounded rectangular opening with three supports. The length is fixed at 600 mm, the width is fixed at 500 mm, and the corner radius is fixed at 200 mm. For more information, see *Rectangular Opening with Three Supports, a 600 mm Length, and a 500 mm Width* (on page 670).
	**3S Rectangle 1350x700x150** - Creates a rounded rectangular opening with three supports. The length is fixed at 1350 mm, the width is fixed at 700 mm, and the corner radius is fixed at 150 mm. For more information, see *Rectangular Opening with Three Supports, a 1350 mm Length, and a 700 mm Width* (on page 675).

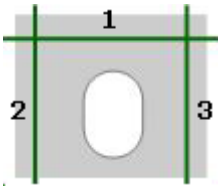
Rectangular Opening with Three Supports



Tooltip: Rect HxWxRxD

Creates a rounded rectangular opening with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports. You define the length, the width, and the corner radius.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

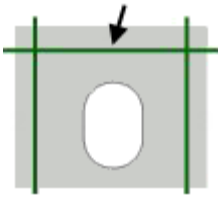
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

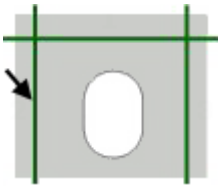
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

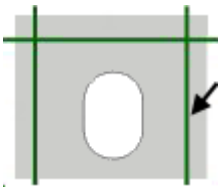
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



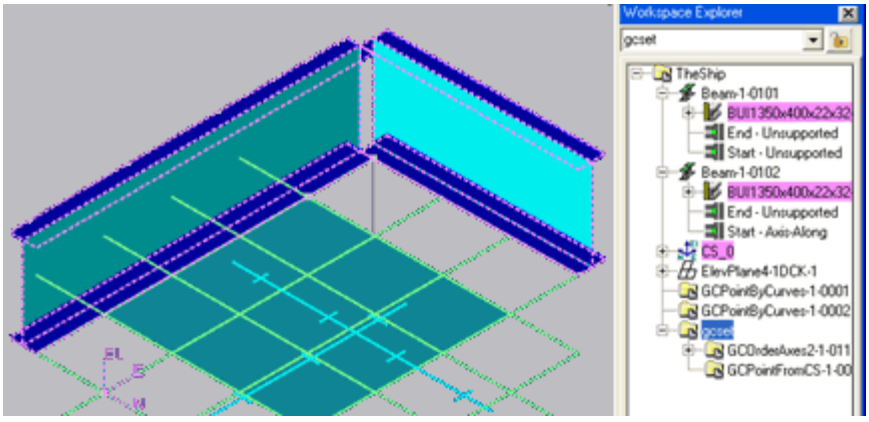
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

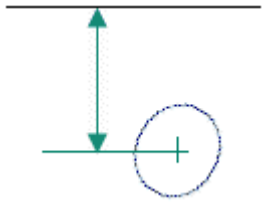
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

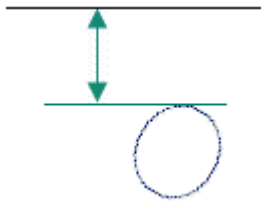
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



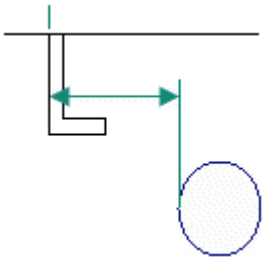
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



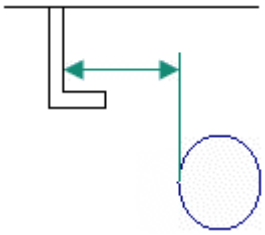
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

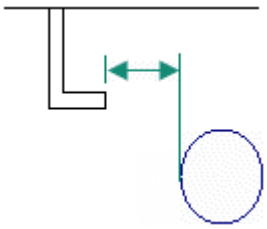
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

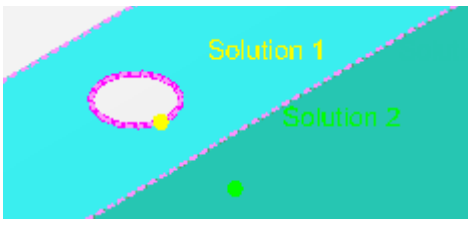
Length - Defines the length (or height) of the rectangular opening.

Width - Defines the width of the rectangular opening.

Rounded Rad - Defines the corner radius of the rounded rectangular opening.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place a rectangular opening with three supports* (on page 658)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

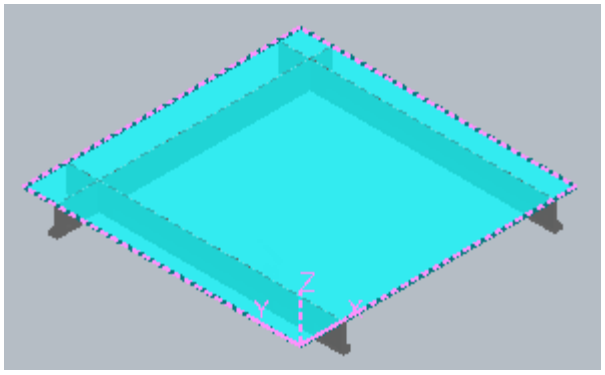
Place a rectangular opening with three supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

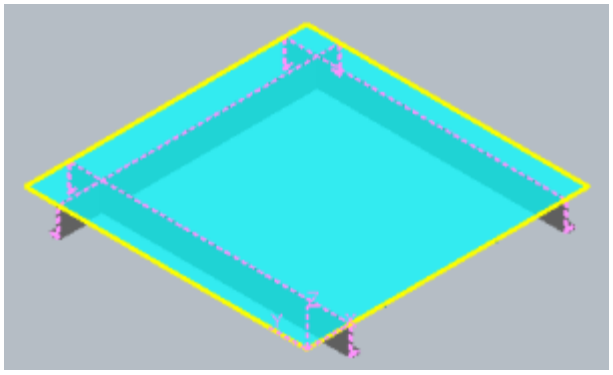
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **3S Rectangle** tab, click **Rect HxWxRxD**.

   The ribbon for the macro displays.
5. Select a coordinate system.

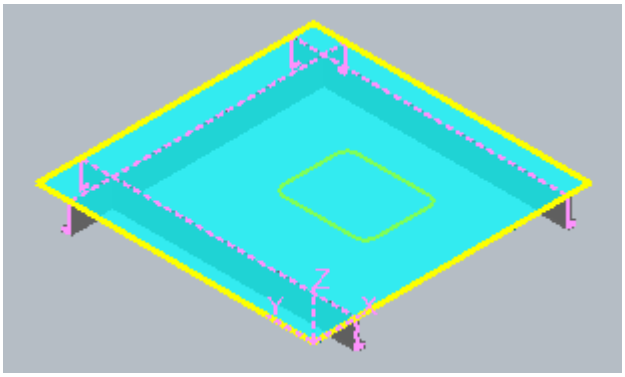
   A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the middle support, and then select the two side supports.



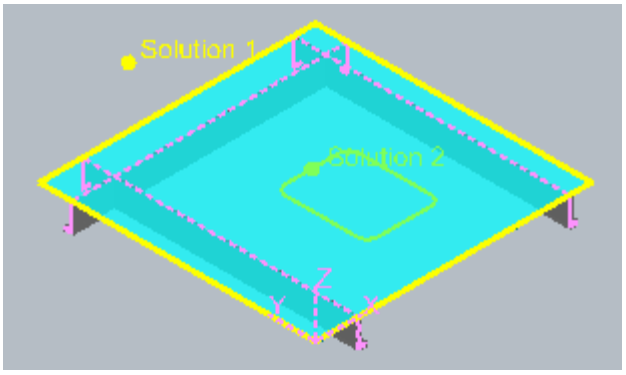
*If the default solution for **Distance Dir** defines geometry on the plate, then a preview displays.*



NOTE *If the default solution for **Distance Dir** does not define geometry on the plate, then a message displays: "Make sure ProjDirection and the DistanceDir are valid and try again."*

8. If a solution is not possible or is not correct:
   a. Click **Distance Dir**.

      The possible solutions display.

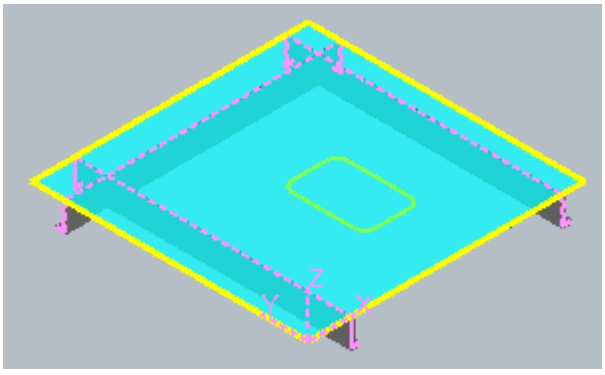


   b. Select the required solution for **Distance Dir**.

      The opening preview displays.

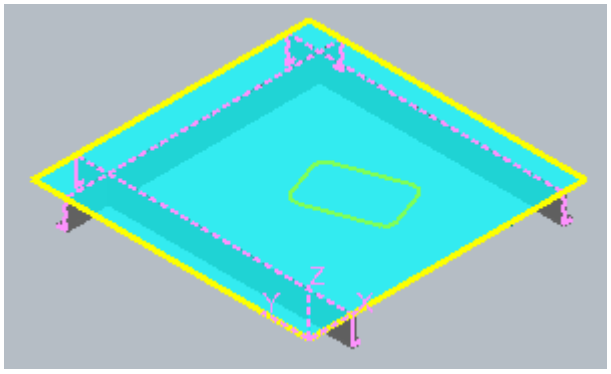
9. If required, type new values for **Length**, **Width**, and **Rounded Rad**.

   The preview reflects the new opening size.



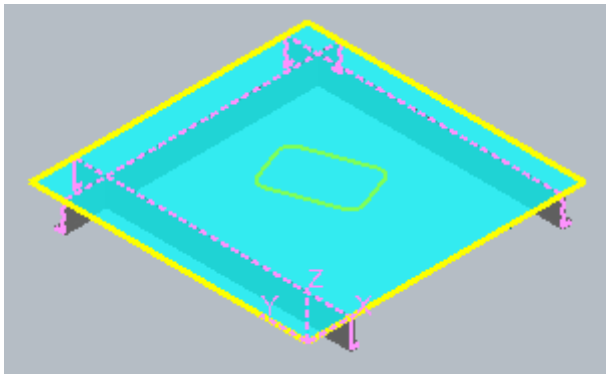
10. If required, type a new value for **Angle**.

    The preview reflects the new angle.



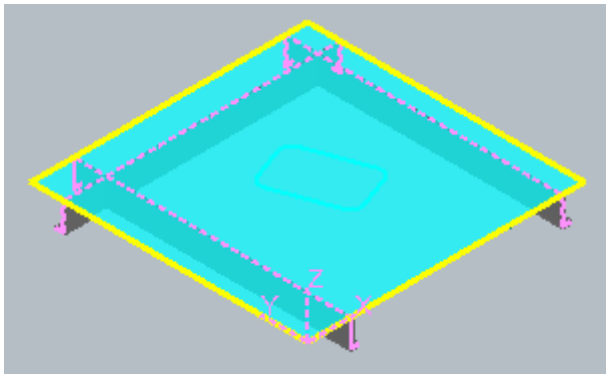
11. If required, change **Distance From** and **Distance** values to change the opening distance with respect to the middle support.

    The preview reflects the new opening location.



12. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

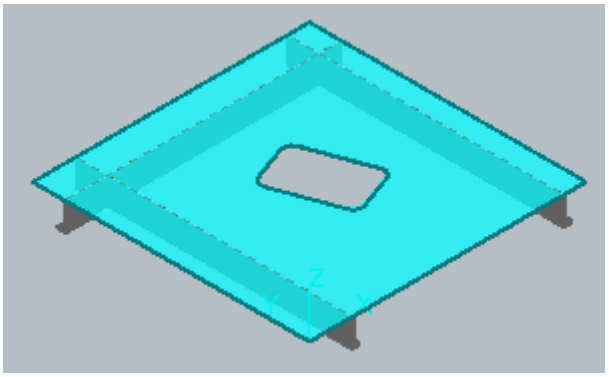
13. Click **Close**.

    *The ribbon display returns to the main **Openings** ribbon.*

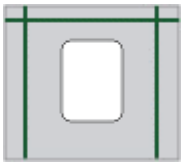
14. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.

15. Click **Finish**.

    The software creates the opening.



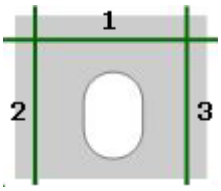
Rectangular Opening with Three Supports, a 400 mm Length, and a 300 mm Width



Tooltip: Rect 400x300x100

Creates a 400 mm by 300 mm rectangular opening with 100 mm radius corners and with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

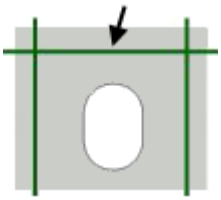
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

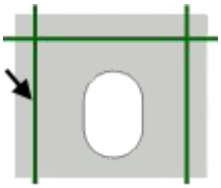
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

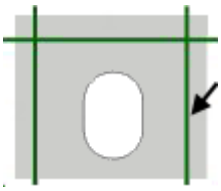
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



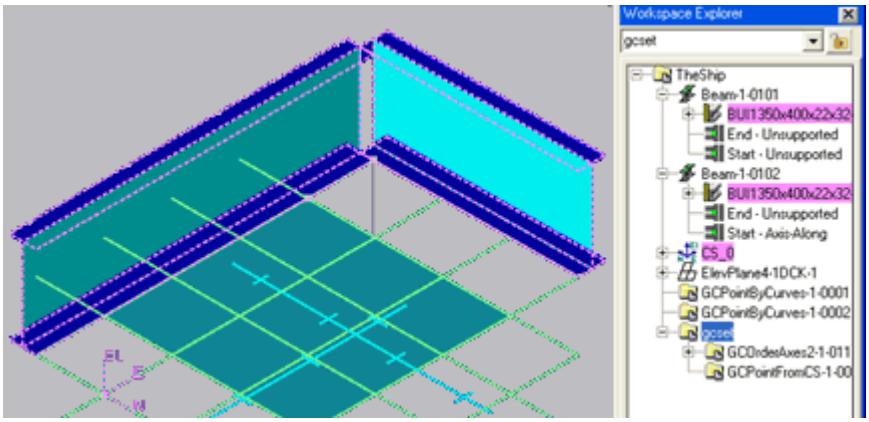
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

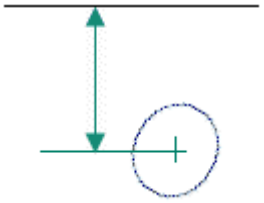
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

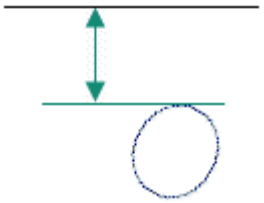
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



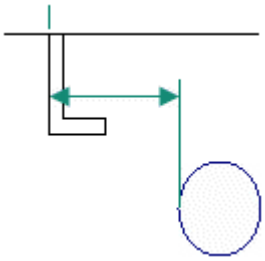
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



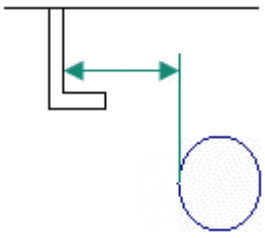
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

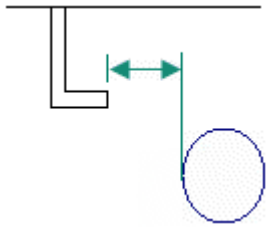
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

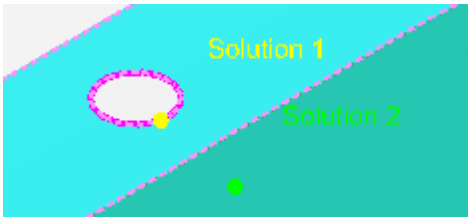
Length - Defines the length (or height) of the rectangular opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the rectangular opening. This option is unavailable and set to the fixed width of the macro.

Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place a rectangular opening with three supports and a fixed height and width* (on page 667)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

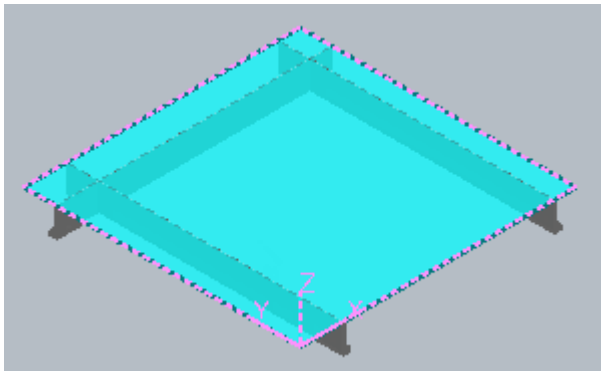
Place a rectangular opening with three supports and a fixed height and width

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

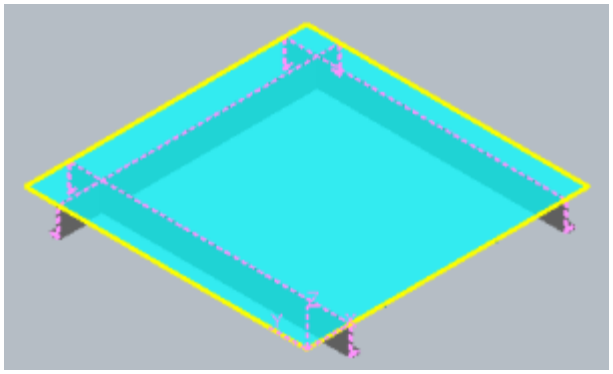
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **3S Rectangle** tab, click **Rect 400x300x100**, **Rect 600x500x200**, or **Rect 1350x700x150**.

   The ribbon for the macro displays.
5. Select a coordinate system.

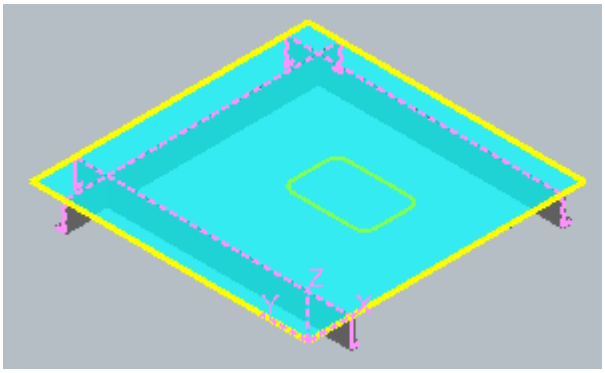
   A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the middle support, and then select the two side supports.



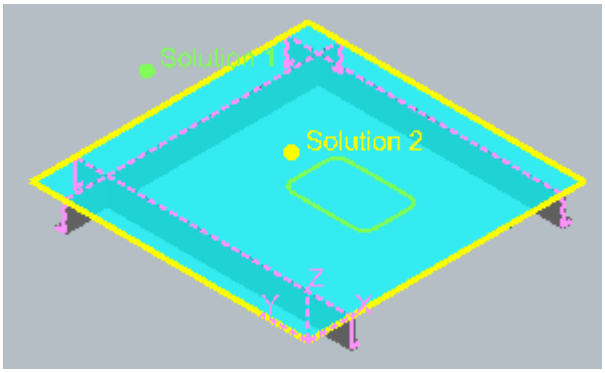
*If the default solution for **Distance Dir** defines geometry on the plate, a preview displays.*



NOTE *If the default solution for **Distance Dir** does not define geometry on the plate, then a message displays: "Make sure ProjDirection and the DistanceDir are valid and try again."*

8. If a solution is not possible or is not correct:
   a. Click **Distance Dir**.

      The possible solutions display.

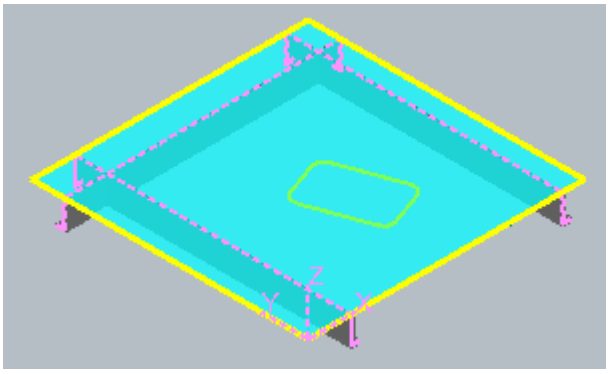


   b. Select the required solution for **Distance Dir**.

      The opening preview displays.

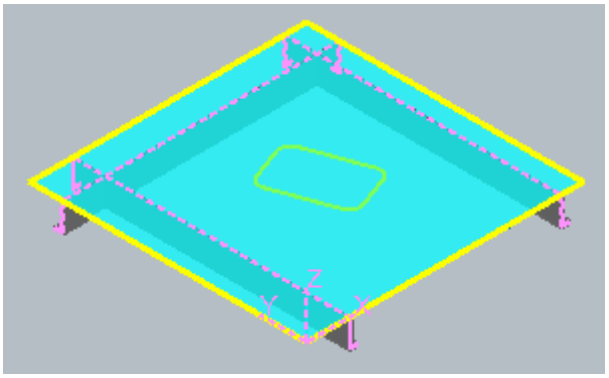
9. If required, type a new value for **Angle**.

   The preview reflects the new angle.



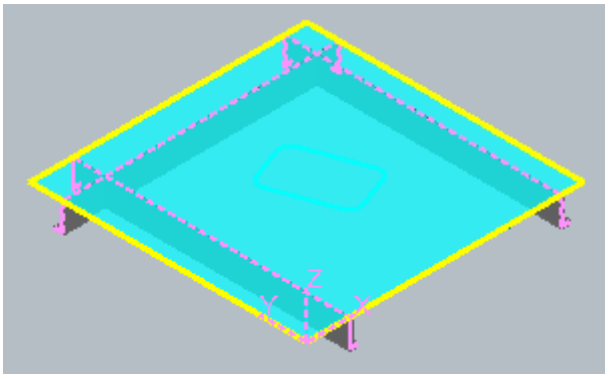
10. If required, change **Distance From** and **Distance** values to change the opening distance with respect to the middle support.

The preview reflects the new opening location.



11. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

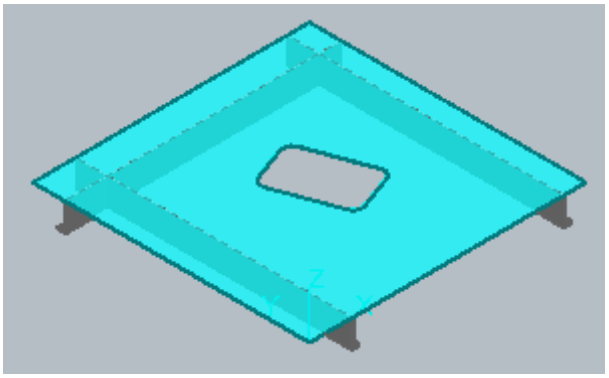
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

12. Click **Close**.

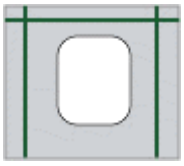
    *The ribbon display returns to the main **Openings** ribbon.*

13. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
14. Click **Finish**.

The software creates the opening.



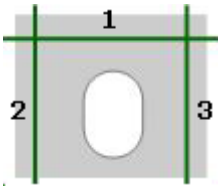
Rectangular Opening with Three Supports, a 600 mm Length, and a 500 mm Width



Tooltip: Rect 600x500x200

Creates a 600 mm by 500 mm rectangular opening with 200 mm radius corners and with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

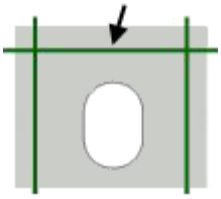
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

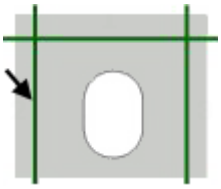
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

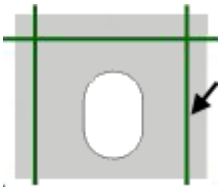
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



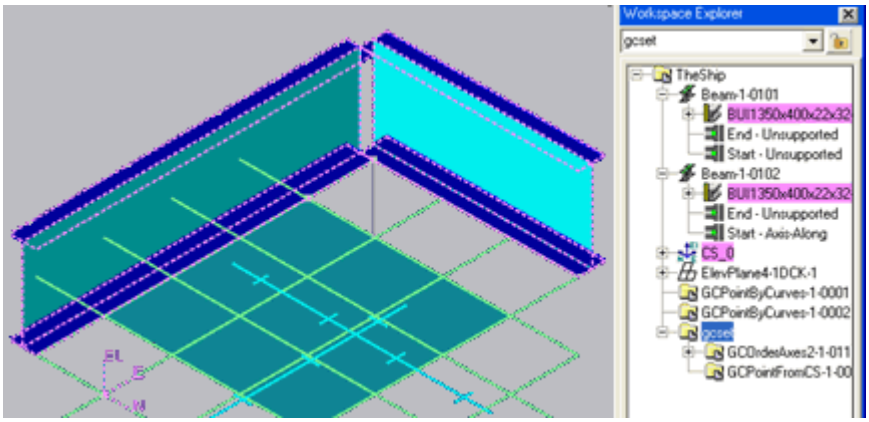
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

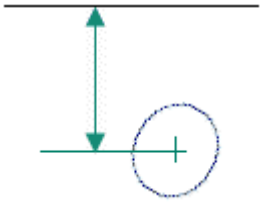
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

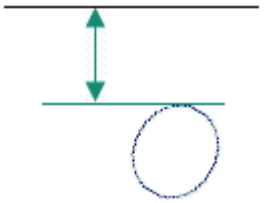
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



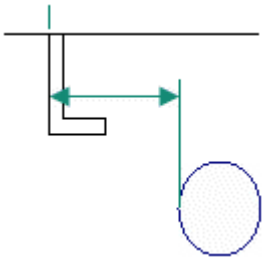
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



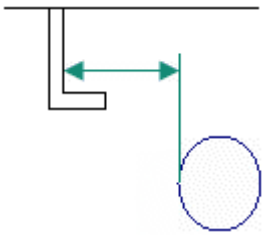
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

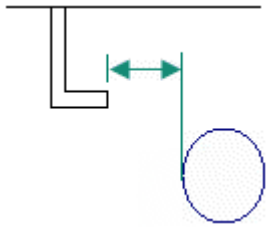
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

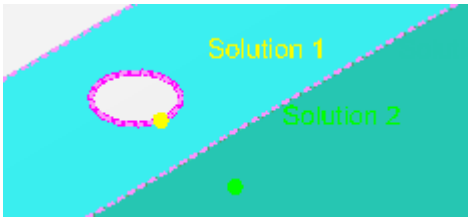
Length - Defines the length (or height) of the rectangular opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the rectangular opening. This option is unavailable and set to the fixed width of the macro.

Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

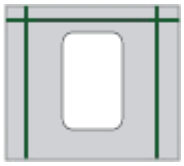
Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.



What do you want to do?

- *Place a rectangular opening with three supports and a fixed height and width* (on page 667)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

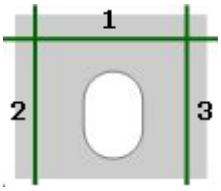
Rectangular Opening with Three Supports, a 1350 mm Length, and a 700 mm Width



Tooltip: Rect 1350x700x150

Creates a 1350 mm by 700 mm rectangular opening with 150 mm radius corners and with three supports. The software positions the opening by a dimension from the first support and is centers the opening between the second and third supports.

Select the middle support first for a 3S opening:



NOTE If opposing supports 2 and 3 are not parallel, then the opening is centered between the midpoint of each support.

3S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

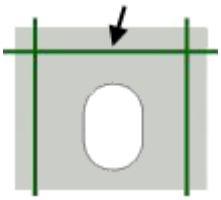
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

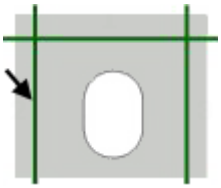
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

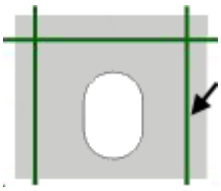
2 - Specifies the middle support used to constrain the opening. A support is a plate system, profile system, or reference plane.



3 - Specifies the first side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



4 - Specifies the second side support used to constrain the opening. A support is a plate system, profile system, or reference plane.



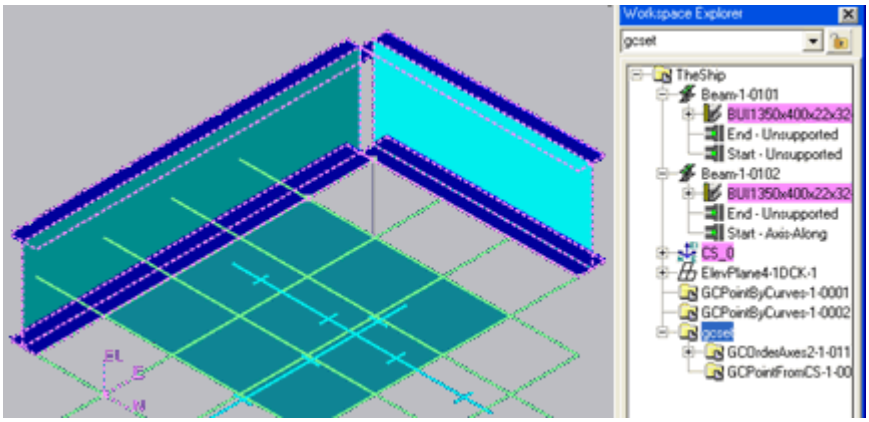
5 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

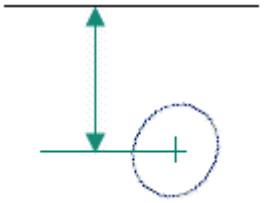
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

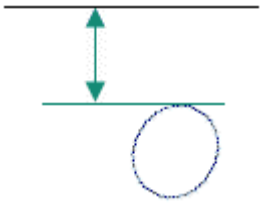
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Distance From - Defines the method used to locate the opening from the first support. Select one of the following options:

- **Center** - Measures from the first support to the center of the opening.



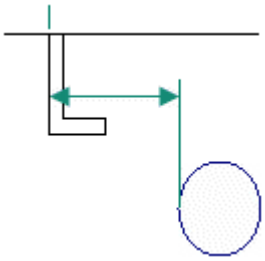
- **Edge** - Measures from the first support to the edge of the opening closest to the support.



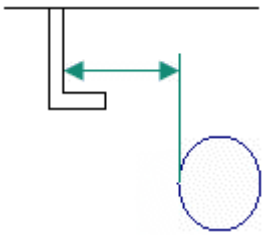
Distance - Defines the distance from the first support to an opening. This distance is defined by the value of **Distance From**.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

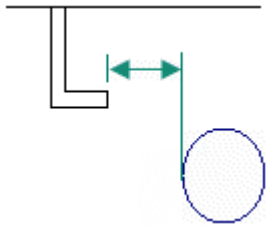
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

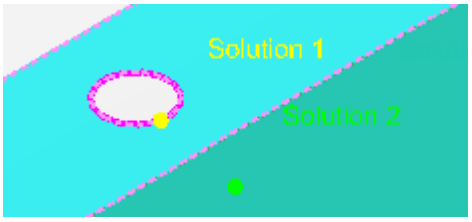
Length - Defines the length (or height) of the rectangular opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the rectangular opening. This option is unavailable and set to the fixed width of the macro.

Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

Distance Dir - Defines the side on which to place the opening if more than one solution is possible. Select **Solution 1** or **Solution 2**. A preview of the opening appears at the selected location.

Select Value in the 3D Graphic View - Displays previews of the possible solutions for **Distance Dir** in the graphic view. Select the solution to use.

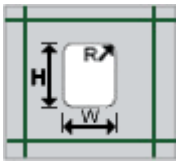
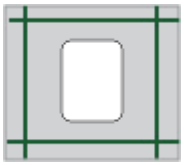
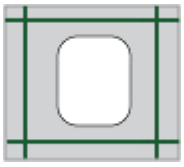
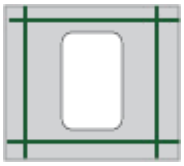


What do you want to do?

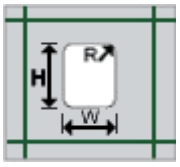
- *Place a rectangular opening with three supports and a fixed height and width* (on page 667)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

4S Rectangle Tab (Opening Palette Dialog Box)

The following advanced opening macros allow you to place rectangular openings with four supports. The software centers a four-support opening between opposing supports.

	**4S Rectangle HxWxR** - Creates a rounded rectangular opening with four supports. You can define the length, the width, and the corner radius. For more information, see *Rectangular Opening with Four Supports* (on page 680).
	**4S Rectangle 400x300x100** - Creates a rounded rectangular opening with four supports. The length is fixed at 400 mm, the width is fixed at 300 mm, and the corner radius is fixed at 100 mm. For more information, see *Rectangular Opening with Four Supports, a 400 mm Length, and a 300 mm Width* (on page 687).
	**4S Rectangle 600x500x200** - Creates a rounded rectangular opening with four supports. The length is fixed at 600 mm, the width is fixed at 500 mm, and the corner radius is fixed at 200 mm. For more information, see *Rectangular Opening with Four Supports, a 600 mm Length, and a 500 mm Width* (on page 694).
	**4S Rectangle 1350x700x150** - Creates a rounded rectangular opening with four supports. The length is fixed at 1350 mm, the width is fixed at 500 700 , and the corner radius is fixed at 150 mm. For more information, see *Rectangular Opening with Four Supports, a 1350 mm Length, and a 700 mm Width* (on page 698).

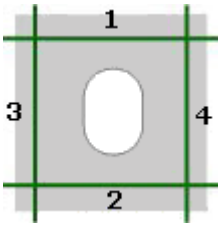
Rectangular Opening with Four Supports



Tooltip: 4SRect HxWxR

Creates a rounded rectangular opening with four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

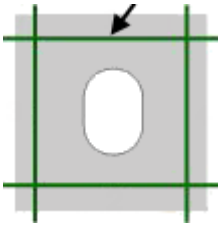
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

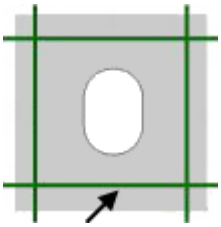
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

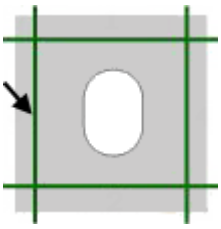
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



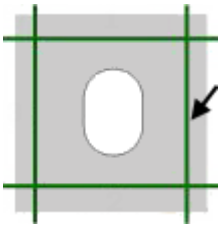
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



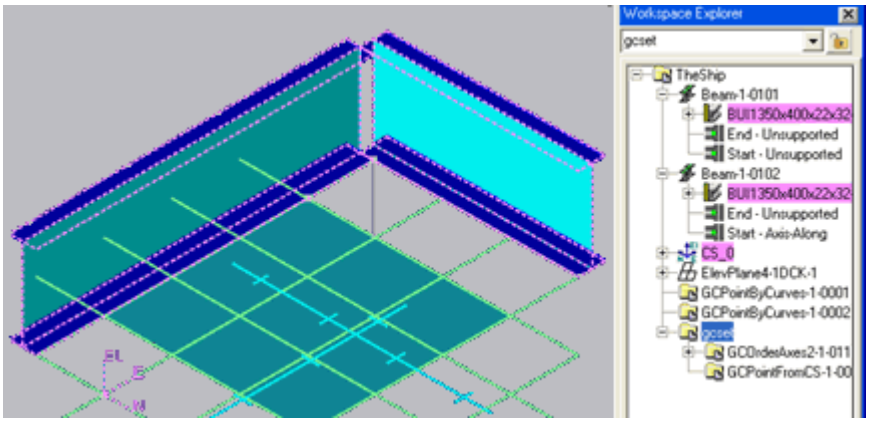
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

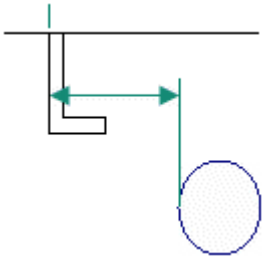
- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

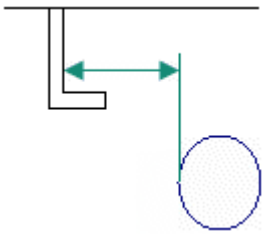
  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

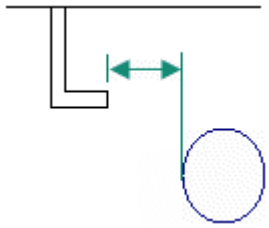
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

Length - Defines the length of the rectangular opening.

Width - Defines the width of the rectangular opening.

Rounded Rad - Defines the corner radius of the rounded rectangular opening.

What do you want to do?

- *Place a rectangular opening with four supports* (on page 684)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

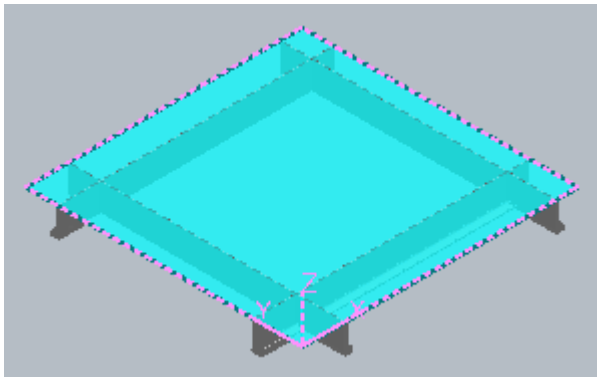
Place a rectangular opening with four supports

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

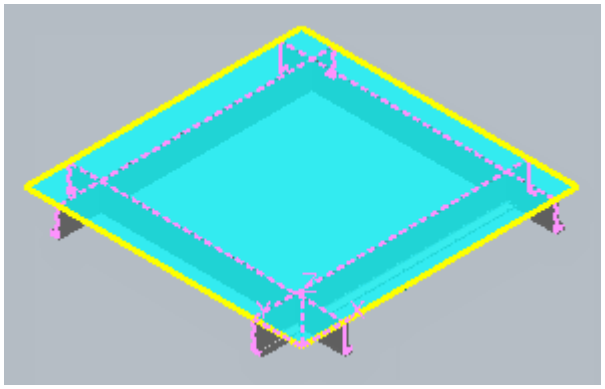
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **4S Rectangle** tab, click **4SRect HxWxR**.

   The ribbon for the macro displays.
5. Select a coordinate system.

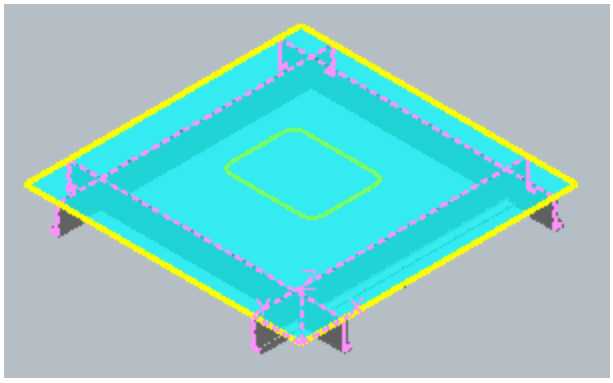
A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the first support, a second support opposite the first support, a third support, and a fourth support opposite the third support.

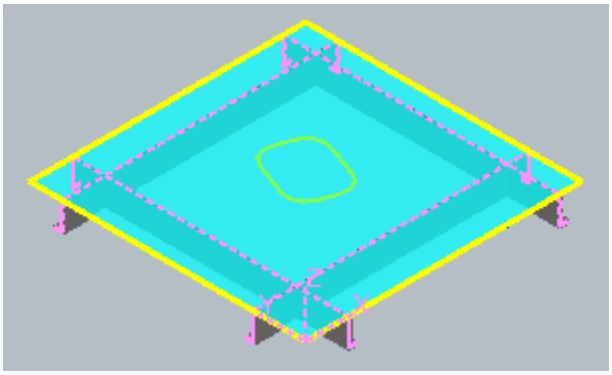


The opening preview displays.



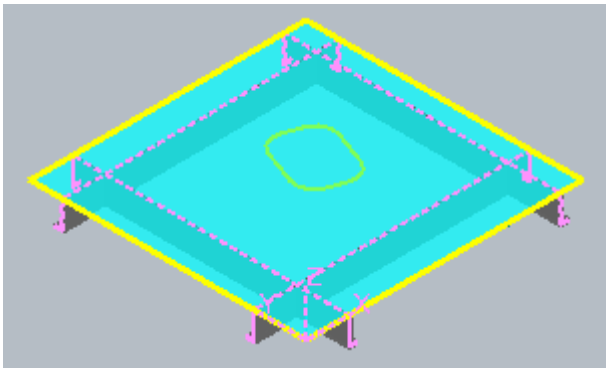
8. If required, type new values for **Angle**, **Length**, **Width**, and **Rounded Rad**.

The preview reflects the new parameters.



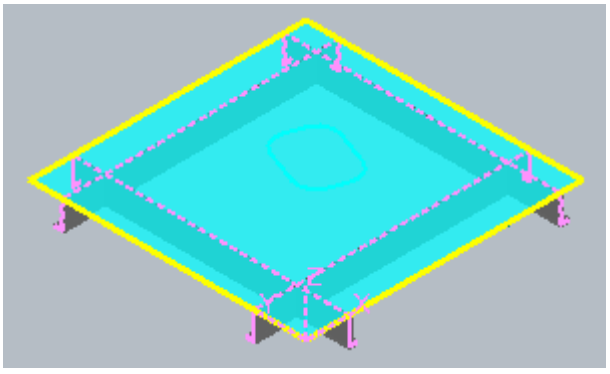
9. If required, change the **Dimension Type** value to change the opening distance from the supports.

   The preview reflects the new opening location.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

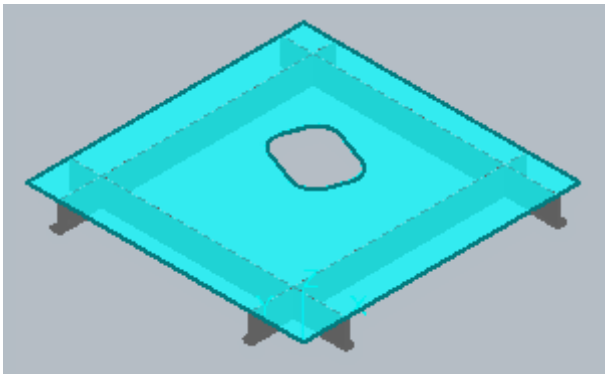
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

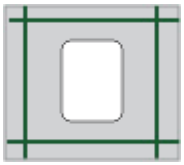
*The ribbon display returns to the main **Openings** ribbon.*

12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

    The software creates the opening.



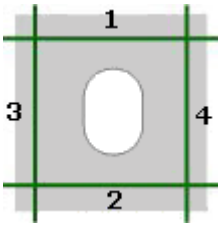
Rectangular Opening with Four Supports, a 400 mm Length, and a 300 mm Width



Tooltip: 4SRect 400x300x100

Creates a 400 mm by 300 mm rectangular opening with 100 mm radius corners and with four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

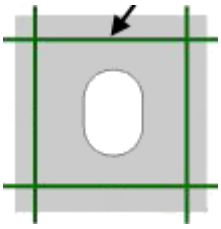
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

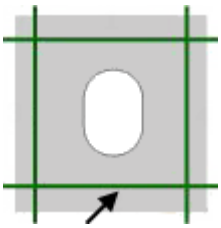
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

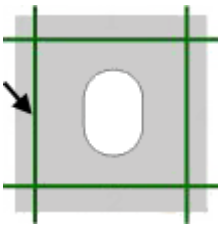
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



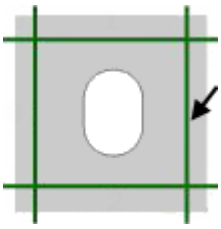
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



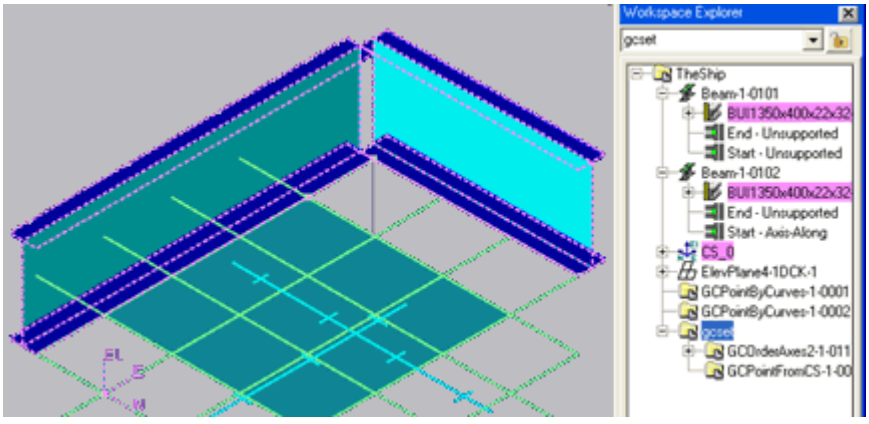
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

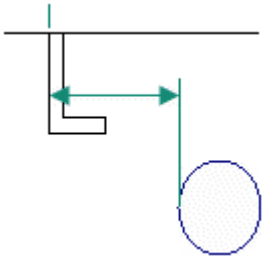
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

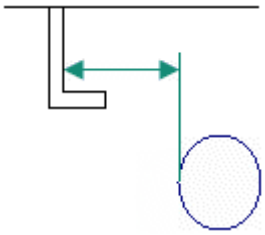
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

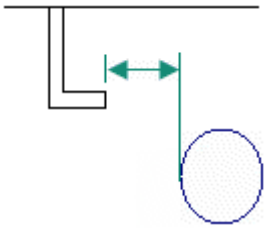
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

Length - Defines the length of the rectangular opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the rectangular opening. This option is unavailable and set to the fixed width of the macro.

Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

- *Place a rectangular opening with four supports and a fixed height and width* (on page 691)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

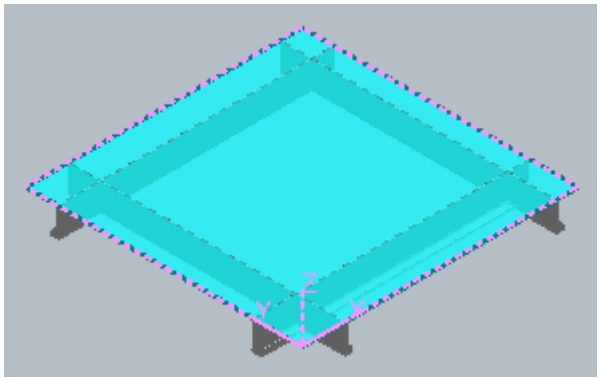
Place a rectangular opening with four supports and a fixed height and width

1. Click **Openings** on the vertical toolbar.
2. Select the plate system on which to place the opening.
3. Click **Define by catalog selection**.

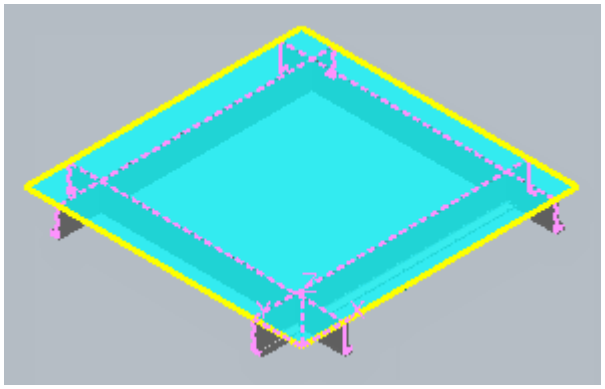
   *The **Opening Palette** dialog box and the **Geometric Construction Explorer** dialog box display.*
4. On the **4S Rectangle** tab, click **4SRect 400x300x100**, **4SRect 600x500x200**, or **4SRect 1350x700x150**.

   The ribbon for the macro displays.
5. Select a coordinate system.

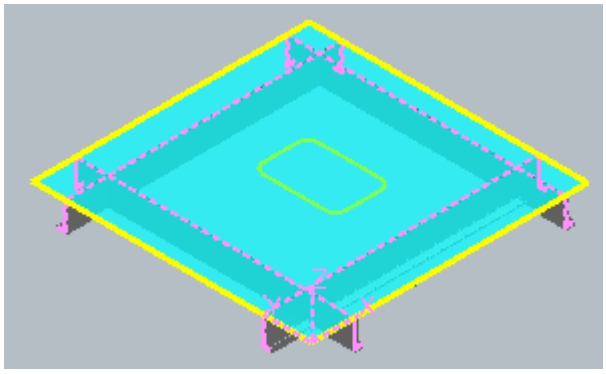
A local coordinate system at the plate origin displays.



6. Click **2**.
7. Select the first support, a second support opposite the first support, a third support, and a fourth support opposite the third support.

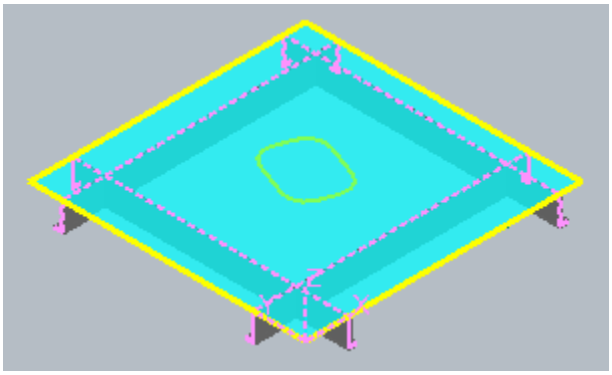


The opening preview displays.



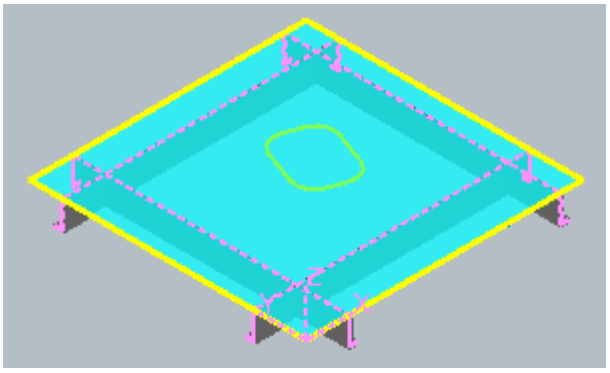
8. If required, type a new value for **Angle**.

The preview reflects the new angle



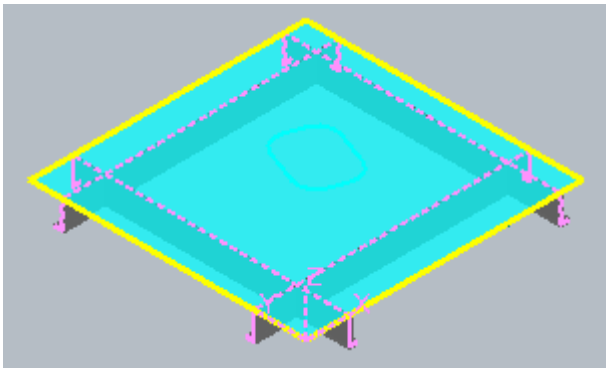
9. If required, change the **Dimension Type** value to change the opening distance from the supports.

   The preview reflects the new opening location.



10. Click **Continue**.

    *The preview colors change. The opening also displays in the **Geometric Construction Explorer** hierarchy.*



TIPS

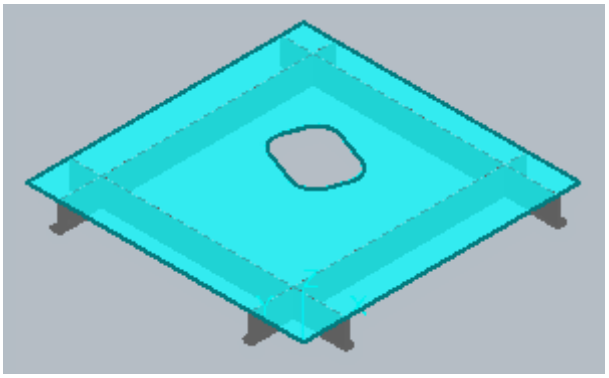
- You can define additional openings before proceeding to the next step. Click **Continue** after defining each opening.
- To delete opening geometry, see *Delete advanced opening geometry* (on page 489).

11. Click **Close**.

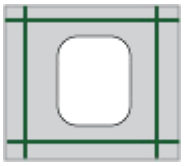
*The ribbon display returns to the main **Openings** ribbon.*

12. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the opening.
13. Click **Finish**.

    The software creates the opening.



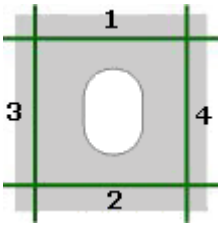
Rectangular Opening with Four Supports, a 600 mm Length, and a 500 mm Width



Tooltip: 4SRect 600x500x200

Creates a 600 mm by 500 mm rectangular opening with 200 mm radius corners and with four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

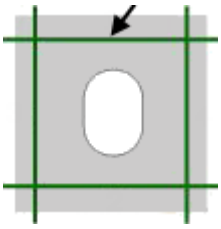
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

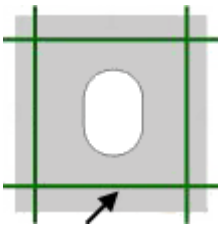
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

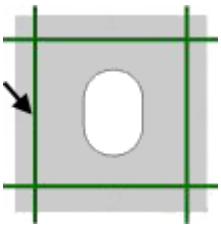
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



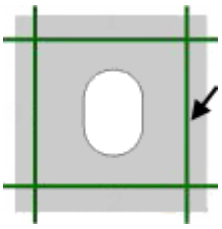
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



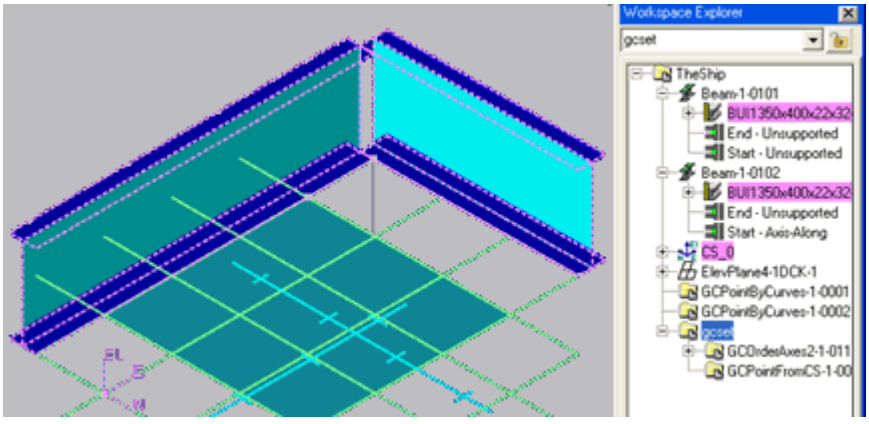
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

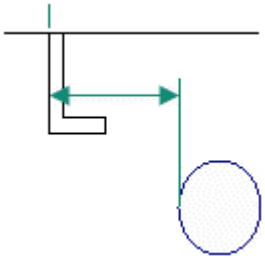
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and does not display when editing an object.

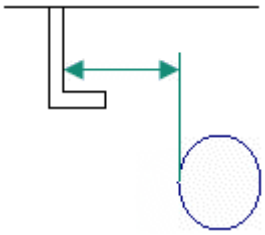
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

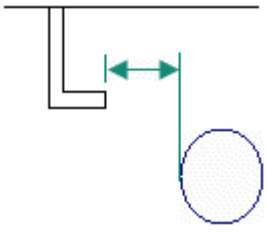
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

Length - Defines the length of the rectangular opening. This option is unavailable and set to the fixed length of the macro.

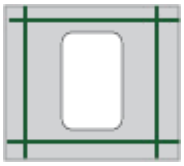
Width - Defines the width of the rectangular opening. This option is unavailable and set to the fixed width of the macro.

Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

What do you want to do?

- *Place a rectangular opening with four supports and a fixed height and width* (on page 691)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

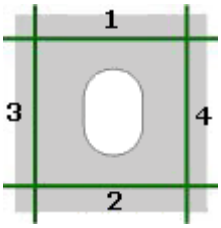
Rectangular Opening with Four Supports, a 1350 mm Length, and a 700 mm Width



Tooltip: 4SRect 1350x700x150

Creates a 1350 mm by 700 mm rectangular opening with 150 mm radius corners and with four supports. The software centers the opening between opposing supports.

Select supports as opposing pairs for a 4S opening:



NOTE If opposing supports 1 and 2, or 3 and 4, are not parallel, then the opening is centered between the midpoint of each support.

4S Fixed Dimension Rectangle Ribbon

Opening Properties - Opens the **Properties** dialog box for the selected opening. This option is only available after selecting an existing opening.

Select - Selects an existing opening object to edit or delete.

Add - Selects geometry that defines a new opening. USe this option with the numbered step options listed below.

Delete - Deletes a selected opening.

Cancel - Cancels the opening creation, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon.

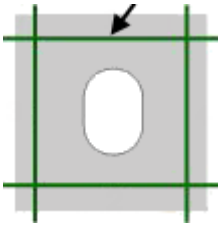
Close - Verifies the option values, closes the ribbon and the **Opening Palette** dialog box, and then returns to the **Openings** ribbon. The selected values display when **Define by catalog selection** is selected again on the **Openings** ribbon.

Geometric Construction - Displays the current opening macro. You can also select the following:

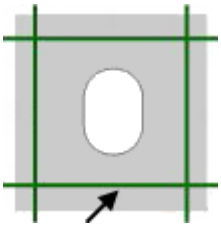
- A different, recently used macro.
- **More** - Opens the **Select Geometric Construction** dialog box. Use this option only for openings that do not appear on the **Opening Palette** dialog box.

1 - Specifies the coordinate system used.

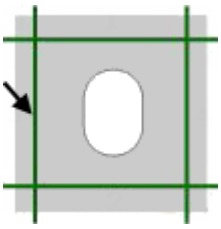
2 - Specifies the first support used to constrain the opening. A support is a plate system, profile system, or reference plane.



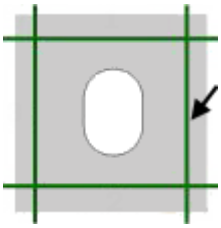
3 - Specifies the second support used to constrain the opening. This support must be opposite of the first support. A support is a plate system, profile system, or reference plane.



4 - Specifies the third support used to constrain the opening. A support is a plate system, profile system, or reference plane.



5 - Specifies the fourth support used to constrain the opening. This support must be opposite of the third support. A support is a plate system, profile system, or reference plane.



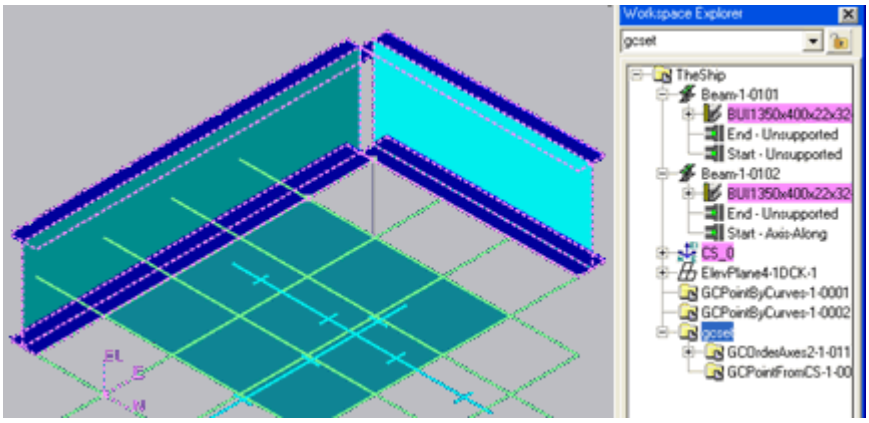
6 - Specifies additional plates on which to create the opening. The plate that you initially selected using the **Plate System** step on the **Openings** ribbon is the default option.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject - Clears the selections for the current step.

Accept - Accepts the current selections and displays a preview.

Continue - Completes the opening definition. Display of the ribbon continues, allowing you to define parameters for additional openings.

★ IMPORTANT

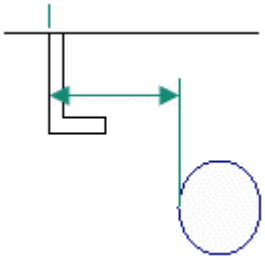
- If you stop the command before clicking **Finish**, the software does not save the opening geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

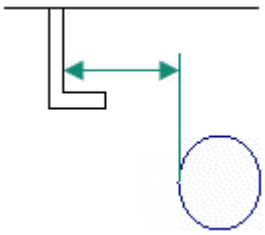
  The message displays only during creation of an object and does not display when editing an object.
- The software does not create opening objects and save to the model until you click **Finish** on the main **Openings** ribbon.

Dimension Type - Defines the measurement location of the support. The edge of the opening is measured from this location. This option is defined separately for each support. Select one of the following options:

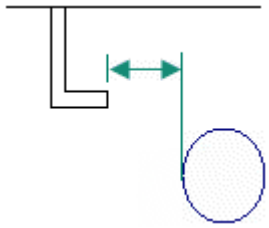
- **Molded** - Measures from the molded side of the support to the edge of the opening.



- **Thickness** - Measures from the thickness side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the web.



- **Clearance** - Measures from the closest side of the support to the edge of the opening. If the support is a profile, then the measurement is from the closest side of the flange.



Projection Direction - Defines the project direction of the opening shape. Select one of the following options:

- **Normal** - Normal to the surface of the plate system.
- **Along X** - In the X direction of the selected coordinate system.
- **Along Y** - In the Y direction of the selected coordinate system.
- **Along Z** - In the Z direction of the selected coordinate system.

Angle - Defines the rotation angle of the opening about the center of the opening. The default value is **0.00 deg**.

Shape - Defines the shape of the opening. This option is unavailable and set to **Rounded Rect**.

Length - Defines the length of the rectangular opening. This option is unavailable and set to the fixed length of the macro.

Width - Defines the width of the rectangular opening. This option is unavailable and set to the fixed width of the macro.

Rounded Rad - Defines the corner radius of the rounded rectangular opening. This option is unavailable and set to the fixed radius of the macro.

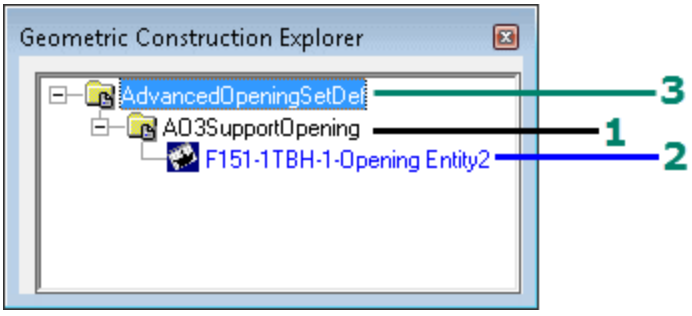
What do you want to do?

- *Place a rectangular opening with four supports and a fixed height and width* (on page 691)
- *Delete advanced opening geometry* (on page 489)
- *Change an advanced opening to a different opening definition* (on page 489)
- *Copy and paste an advanced opening* (on page 490)

Geometric Construction Explorer

Displays geometry related to an opening. The geometry is added after opening parameters are defined and **Continue** is clicked on the **Advanced Openings** ribbon.

The **Geometric Construction Explorer** displays parameters in a hierarchical list similar to the Workspace Explorer. Selecting a blue opening geometry name highlights the geometry in the graphic view.



1 - Opening macro (black text)

2 - Output opening geometry (blue text)

3 - Geometric construction set

Commands

The following commands are available by right-clicking an item in the hierarchical list:

Show

Turns on the graphical display of a hidden object in the graphic view.

Hide

Turns off the graphical display of an object in the graphic view. The names of hidden objects display in italics in the **Geometric Construction Explorer**.

Export

Exports the selected geometric construction to an XML file. This command is only available when you right-click the top level geometric construction set.

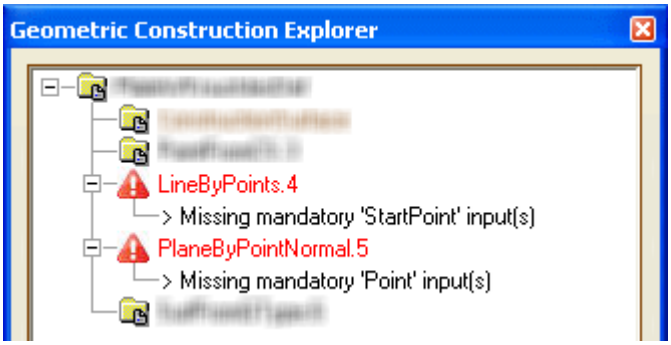
Import

Imports a previously exported geometric construction from an XML file into the workspace. The imported geometric construction replaces any existing geometric construction. This command is only available when you right-click the top level geometric construction set.

NOTE **Export** and **Import** are primarily intended for use by administrators creating complex geometric construction macros. The commands can also be used to save multiple geometric constructions, such as a set of seams or profile stiffener landing curves on a hull, that you may not complete in a single session, or that you want to reuse in a similar model.

Error Display

When a required input for a geometric construction is deleted, the **Geometric Construction Explorer** displays an error, such as those shown in the following example.

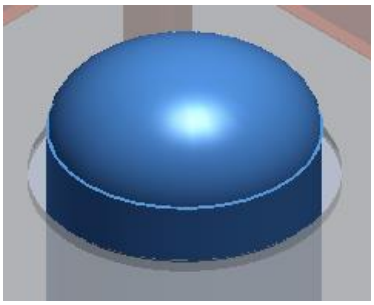


Deleting Opening Geometry

The **Geometric Construction Explorer** is best used to delete opening geometry when the **Advanced Openings** ribbon is open and before openings are created. In this way, you can create geometry for multiple openings, delete geometry as needed, and then create all of the openings at one time by clicking **Finish** on the main **Openings** ribbon.

SECTION 5

Place Opening



Places an opening (hole) in a slab, wall, or member created in the Structure or Civil task. The shape of the opening can be defined by placing a pre-defined shape from the catalog, sketching the opening outline, or defining the boundaries for the opening. In addition, you can control the depth of the opening to create a fully penetrating hole or a recessed opening.

NOTE This command is named **Member Openings** in the Molded Forms task and is typically used with members created with **Place Linear Member System**.

Place Opening Ribbon

Specifies the properties for the opening that you are placing or editing.



Opening Properties

Activates the **Opening Properties** dialog box. You can use this dialog box to specify opening properties that you cannot set on the ribbon. For more information, see *Opening Properties Dialog Box* (on page 719).

Structure

Select the object in which to place an opening. You can select members or slabs.

Sketching Plane

Select the plane on the object on which to draw the opening. If you are placing an opening that does not go all the way through the object, the sketching plane defines the side of the object in which the partial opening is placed.

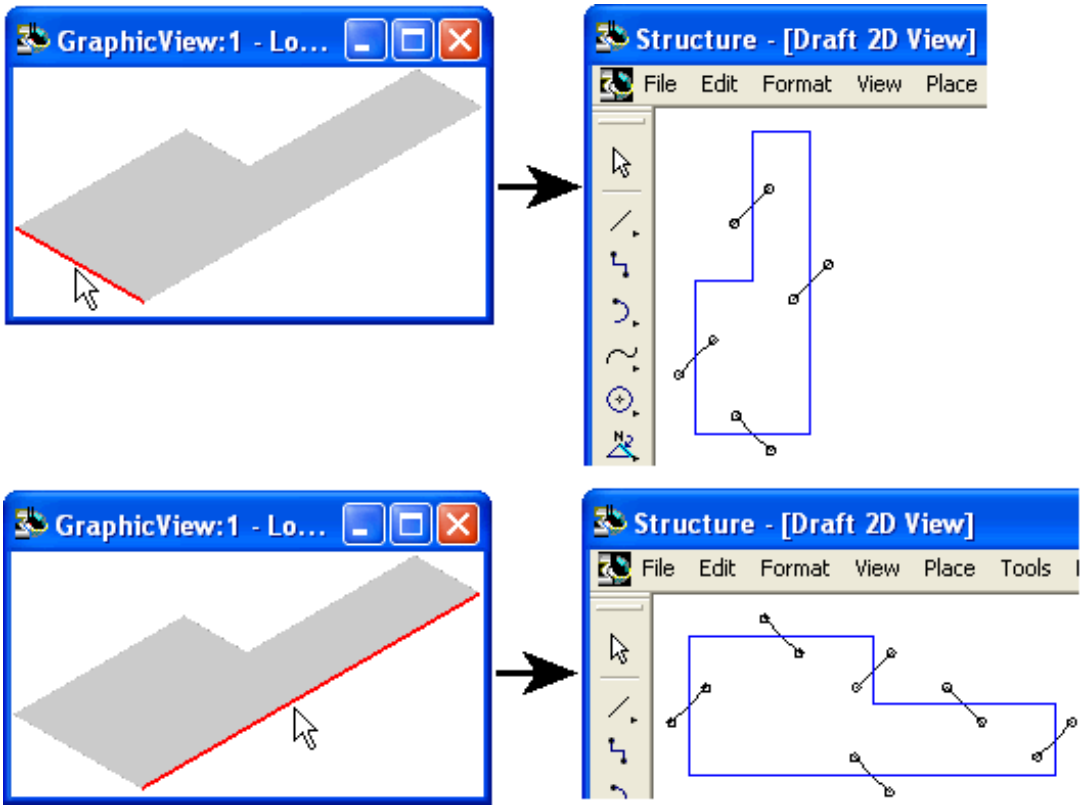
NOTE If you are editing an opening in the Structure or Civil task, this option is only available if the sketching plane for the opening is missing.

Define Orientation

Select this option if you plan to sketch the opening and want to define the orientation of the object, in which the opening is being placed, in the 2D environment. Most of the time, the default orientation that the software uses is appropriate for the opening. However, use this option if you are placing an opening in a multi- sided slab, for example, and you need to know that the correct side of the slab is selected for the opening.

Edge for 2D X-axis

Select this option to specify the object edge that should be oriented along the bottom of the 2D environment. This option is only available when you select the **Define Orientation** option.



End Point

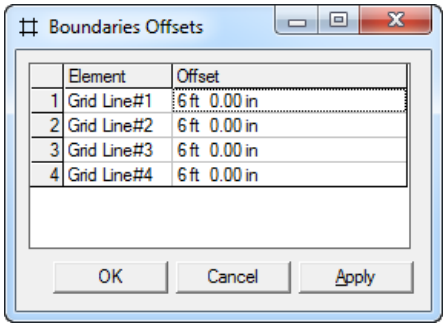
Select this option to specify the lower left corner of the object as you want it to appear in the 2D environment. This option is only available when you select the **Define Orientation** option.

Boundary

Specify the boundaries of the opening. If you select objects in the model and those objects are moved, the software automatically resizes the opening to maintain the boundary relationship. For example, you can select the outside edge of a beam flange as a boundary along one side of an opening. Later, if the section size for the beam is changed and the flange edge moves out an inch, the software automatically moves the edge of the opening an inch to match the new location of the flange edge.

Boundary List

Activates the **Boundary Offsets** dialog box. This dialog box allows you to define an offset distance for each boundary that you have defined. Select a row to highlight the boundary in the model. Then, type the distance from the boundary to place the edge of the opening. Type 0 to place the opening edge on the boundary. Type a negative number to offset the opening inside the boundary. Type a positive number to offset the opening outside the boundary.



Finish

Places the opening using the defined parameters.

Cancel

Rejects the objects that you have selected.

Accept

Confirms the objects that you have selected.

Select Boundaries

Select this option to select objects in the model to define the boundaries of the opening. This option is only available when you are specifying boundaries.

Add References to Sketch 2D

Adds an object as a reference object in the 2-D environment. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model. This option is only available when you are drawing the boundaries of the opening.

NOTE For piping and equipment objects in plant mode, **Insulation**, **Equipment Hole**, **Maintenance**, and **Operation** aspects displaying in the 3D graphic view also display in the Sketch 2D environment. The aspect geometry displays as dashed lines. You can define dimensions, but not relationships, to these aspects in 2D.

TIP The software also automatically retrieves 3-D objects to display in the 2-D environment. The automatically selected objects display as fine grey lines. Examples of these objects include geometry generated by end cut operations, geometry that represents hidden lines from intersecting objects projected to the sketching plane, or objects that intersect the sketching plane. The software writes these graphics to the No_Constrained_Elements layer. You can reference these objects to place your sketch, but the objects that display as fine grey lines are temporary. When you exit the 2-D environment,

the software deletes these graphics. Any relationships or constraints to the temporary graphics are also deleted when you exit the 2-D environment.

Although the references are deleted when you exit the 2D environment, the references are automatically reimported when you modify the opening.

Draw

Activates the Structure Draft 2D View environment. Use this environment to sketch the outline of the opening in the structure that you have selected.

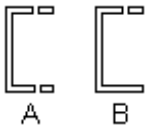
Shape

Specifies how you want to define the opening shape. Select **Sketch** to draw the opening shape. Select **More** to select an opening shape from the catalog. This option is only used when you are not using boundaries to define the opening shape.

Cutting Limit

Select the cutting depth method for the opening.

- Select **Through-All** to completely penetrate the object in which the opening is placed. This option is available for all objects in which you can place an opening.
- Select **Through-Next** to penetrate only one side of an object. For example, a channel section member in which you want an opening only on the top flange. This option is available only when the object that you select for the opening is a member whose cross-section has parallel webs or flanges or a wall that turns back on itself. In the figure, A is the **Through-All** option and B is the **Through-Next** option.



- Select **User Defined** if you want to specify the penetration depth of the opening. This option is available for all objects in which you can place an opening.

Cutting Depth

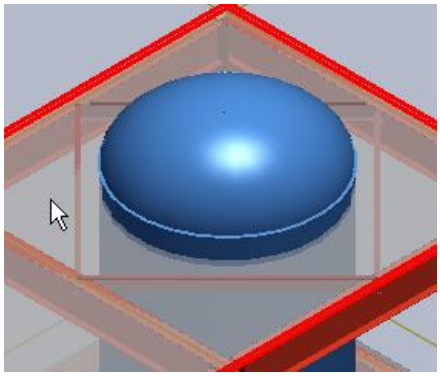
Specify the cutting depth for the opening. The depth is measured from the sketching plane that you specified.

What do you want to do?

- *Place an opening by boundaries* (on page 709)
- *Place an opening by shape* (on page 710)
- *Place an opening by drawing* (on page 711)
- *Place an opening on a sloped wall* (on page 713)
- *Place an opening on a leaning wall* (on page 715)
- *Change a recess opening to a through opening* (on page 717)
- *Change a through opening to a recess opening* (on page 717)
- *Delete an opening* (on page 717)

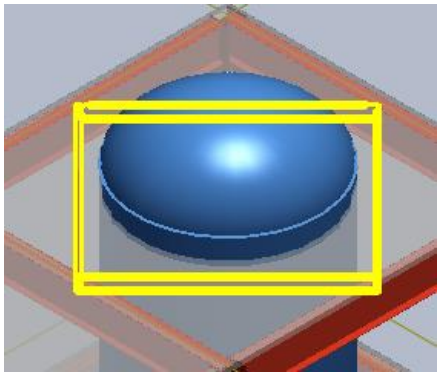
Place an opening by boundaries

1. Click **Place Opening** on the vertical toolbar.
2. Select the object in which you want to place the opening. You can select walls, slabs, and members.

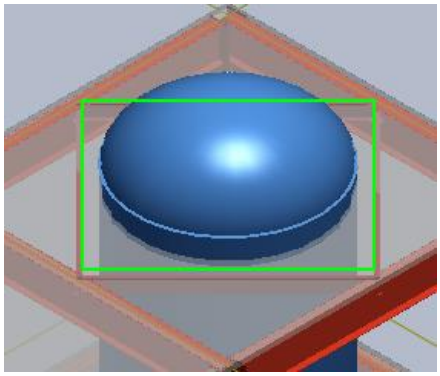


3. Click **Select Boundaries**.
4. Select objects in the model to use as boundaries of the opening.

NOTE You must select the boundary objects in a clockwise or counterclockwise order.



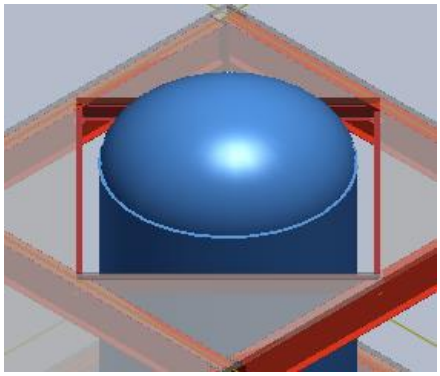
5. Click **Accept**.



6. Optionally, define any offset from the boundaries that you have selected.

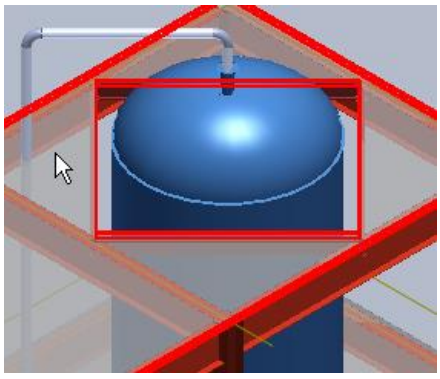
TIP Type 0 to place the opening edge on the boundary. Type a negative number to offset the opening inside the boundary. Type a positive number to offset the opening outside the boundary. This offset can only be applied when the selected boundaries are lines or planes.

7. Click **OK** on the **Boundary** dialog box.
8. Click **Finish**.

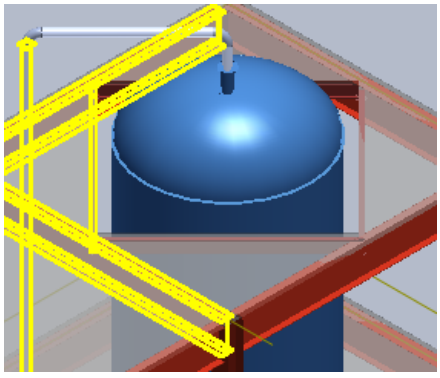


Place an opening by shape

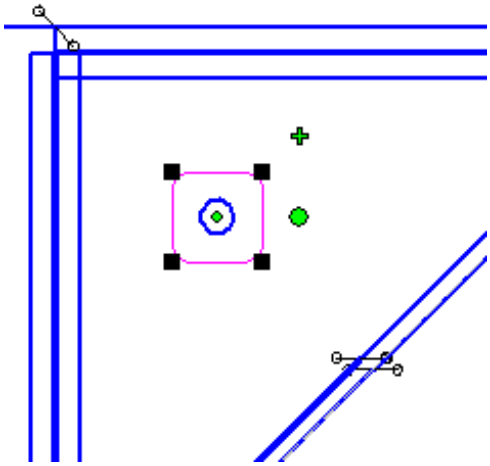
1. Click **Place Opening** on the vertical toolbar.
2. Select the object in which to place the opening. You can select walls, slabs, and members.



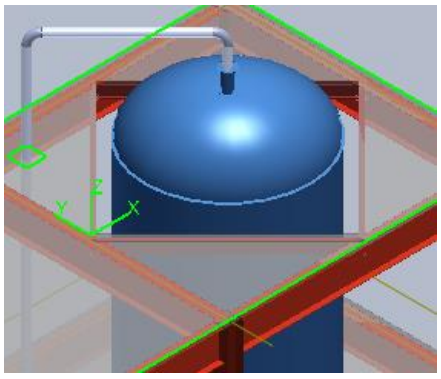
3. Click **Add References to Sketch 2D**.
4. Select objects that you need to reference when placing the opening.



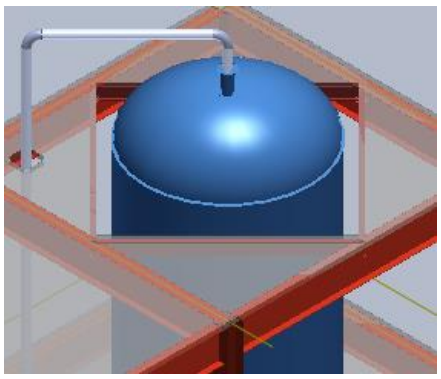
5. In the **Shape** option, select **More...**.
6. Select the catalog shape that you want to place, and then click **OK**.
7. Drag the shape to where you want the opening on the object.



8. Click **Close** on the ribbon bar.



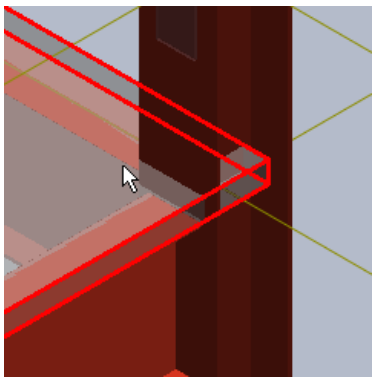
9. Click **Finish**.



Place an opening by drawing

1. Click **Place Opening** on the vertical toolbar.

2. Select the object in which to place the opening. You can select walls, slabs, and members.

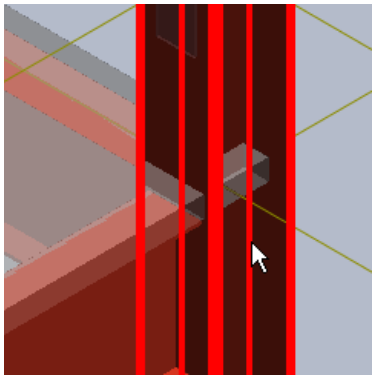


3. Click **Add References to Sketch 2D** .

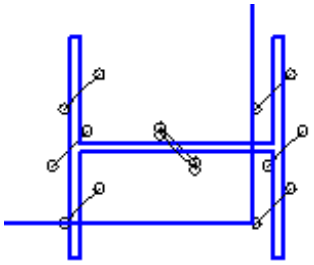
4. Select objects in the 3-D model near the object in which you are placing the opening. The objects that you select display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D.

   TIP The software also automatically retrieves 3-D objects to display in the 2-D environment. The automatically selected objects display as fine grey lines. Examples of these objects include geometry generated by end cut operations, geometry that represents hidden lines from intersecting objects projected to the sketching plane, or objects that intersect the sketching plane. The software writes these graphics to the No_Constrained_Elements layer. You can reference these objects to place your sketch, but the objects that display as fine grey lines are temporary. When you exit the 2-D environment, the software deletes these graphics. Any relationships or constraints to the temporary graphics are also deleted when you exit the 2-D environment.

   Although the references are deleted when you exit the 2D environment, the references are automatically reimported when you modify the opening.

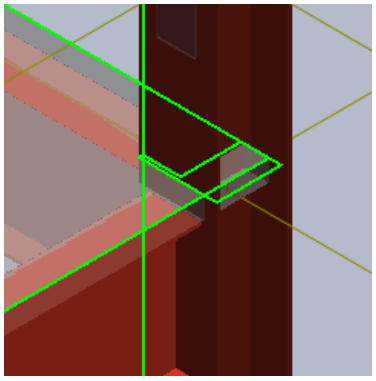


5. Click **Draw** .

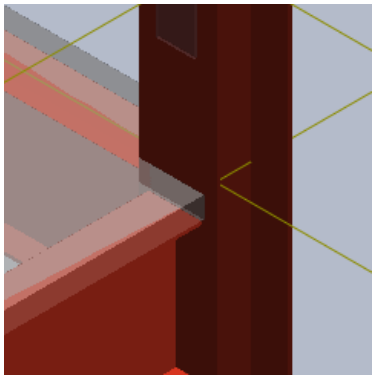


6. Using the available drawing commands in the 2-D environment, draw the opening.
7. Click **Close** on the ribbon bar.

   The opening that you drew displays in the 3-D environment.



8. Click **Finish**.



Place an opening on a sloped wall

Hole Parallel with Wall Slope

1. Click **Place Opening** on the vertical toolbar.
2. Select the sloped wall.
3. Select the side of the wall in which to place the opening.

TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.

4. Click **Draw** .

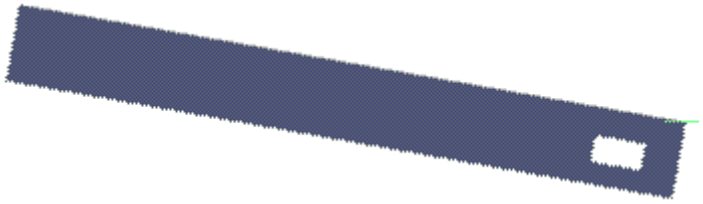
   TIP Regardless of the slope, the wall displays as level in the 2-D environment.

5. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, a small rectangular hole was drawn at the wall end.



6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

   The software places the hole in the sloped wall parallel to the slope.



Hole Parallel with Another Model Object

1. Click **Place Opening** on the vertical toolbar.
2. Select the sloped wall.
3. Select the side of the wall in which to place the opening.

   TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.

4. Click **Add References to Sketch 2D** .
5. Select an object in the 3-D model near the wall that you want the hole parallel to. For example, select a grid line to make the hole parallel to a plane. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model.

   TIP The software also automatically retrieves 3-D objects to display in the 2-D environment. The automatically selected objects display as fine grey lines. Examples of these objects include geometry generated by end cut operations, geometry that represents hidden lines from intersecting objects projected to the sketching plane, or objects that intersect the sketching plane. The software writes these graphics to the

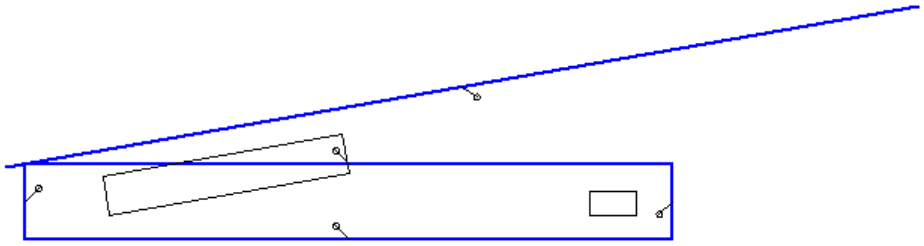
No_Constrained_Elements layer. You can reference these objects to place your sketch, but the objects that display as fine grey lines are temporary. When you exit the 2-D environment, the software deletes these graphics. Any relationships or constraints to the temporary graphics are also deleted when you exit the 2-D environment.

Although the references are deleted when you exit the 2D environment, the references are automatically reimported when you modify the opening.

6. Click **Draw** .

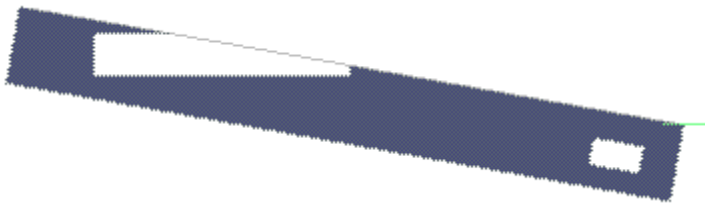
   The wall and the selected object displays in the 2-D environment.

7. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, the opening was placed using the parallel SmartSketch relationship with a grid line that was selected as a reference.



8. Click **Close** on the ribbon bar.
9. Specify the cutting limit to use.
10. Click **Finish**.

    The software places the hole in the sloped wall parallel to the referenced grid line.



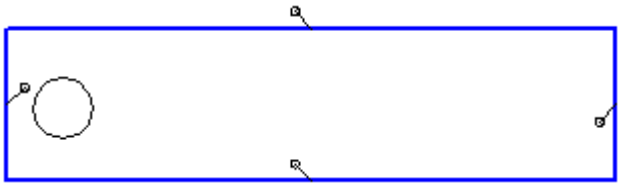
Place an opening on a leaning wall

Opening Normal to Side of Wall

1. Click **Place Opening** on the vertical toolbar.
2. Select the wall.
3. Select the side of the wall in which to place the opening.

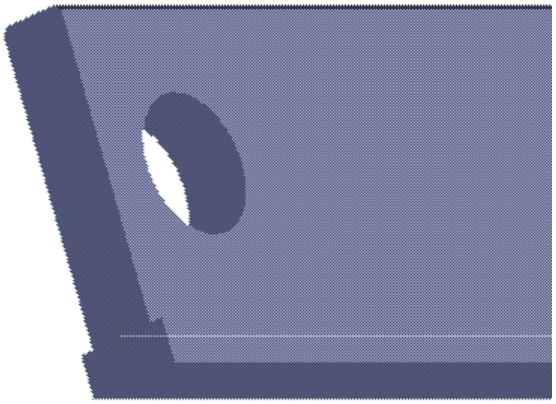
   TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.

4. Click **Draw** .
5. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, a circular hole was drawn at the wall end.



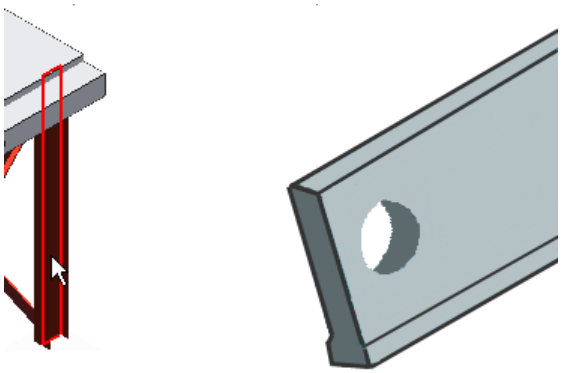
6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

   The software places the opening normal to the wall face.



Opening Normal to Another Model Object

1. Click **Place Opening** on the vertical toolbar.
2. Select the wall.
3. Select a plane that is normal to the opening that you want. In the example below, the web of a nearby column is selected.

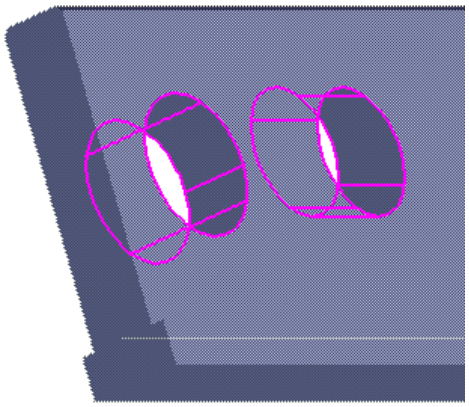


4. Click **Draw** .

The wall displays in the 2-D environment.

5. Using the available drawing commands in the 2-D environment, draw the opening.
6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

   The software places the opening normal to the column's web (right opening), not the wall face (left opening).



Change a recess opening to a through opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the recess opening to change to a through opening.
4. In the **Cutting Limit** option, select **Through-All**.

Change a through opening to a recess opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the through opening to change to a recess opening.
4. In the **Cutting Limit** option, select **User Defined**.
5. In the **Cutting Depth** option, specify the new depth of the opening.

Delete an opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the opening to delete.

4. Click **Delete** .

Opening Properties Dialog Box

Specifies the properties for the opening that you are editing.

See Also

General Tab (Opening Properties Dialog Box) (on page 719)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
Notes Tab (on page 817)

General Tab (Opening Properties Dialog Box)

The **General** tab displays the opening properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one opening and then selected the properties command, only the common properties between the selected openings display.

When viewing properties for a single opening, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category - Select the properties that you want to view for the opening. Opening properties have one category: Standard. You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the opening. Because opening reference data is fully customizable, only the common properties are documented.

Name

Specify a name for the opening.

Naming Rule

Select the naming rule to use to name the opening.

- **Default Name Rule** - Names the opening using the format "Opening-<location>-<index>" where <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, Opening-1-0043.
- **Unique Name Rule** - Names the wall using the format "Opening-<location>-<index>" where <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, Opening-1-0043.
- **User Defined** - Select to specify the wall system name yourself in the **Name** box.

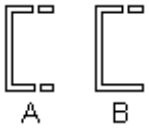
Description

Type a description for the opening.

Cutting Limit

Select the cutting depth method for the opening.

- Select **Through-All** to completely penetrate the object in which the opening is placed. This option is available for all objects in which you can place an opening.
- Select **Through-Next** to penetrate only one side of an object. For example, a channel section member in which you want an opening only on the top flange. This option is available only when the object that you select for the opening is a member whose cross-section has parallel webs or flanges or a wall that turns back on itself. In the figure, A is the **Through-All** option and B is the **Through-Next** option.



- Select **User Defined** if you want to specify the penetration depth of the opening. This option is available for all objects in which you can place an opening.

Cutting Depth

Specify the cutting depth for the opening. The depth is measured from the sketching plane that you specified.

Default constant offset

Specify the offset between the opening edge and the boundary. Type 0 to place the opening edge on the boundary. Type a negative number to offset the opening inside the boundary. Type a positive number to offset the opening outside the boundary. This offset can only be set when the boundaries are lines or planes.

See Also

Opening Properties Dialog Box (on page 719)

SECTION 6

Placing Profile Systems

The Profile System commands define a single profile system or multiple profile systems. Commands are available for stiffener profile systems, linear beam profile systems, edge reinforcements, and tripping stiffener systems. Additional commands from the Structure task are also available for creating members and cans.



Five methods are available for creating the landing curves of stiffener profile systems:

- By intersection
- By offset
- By two-dimensional sketch projection
- By entering coordinates in a table
- By selecting a starting object, an ending object, and an orientation (for a tripping stiffener)

When you create a landing curve by one of these methods, it can be linear, curved, polyline, circle, circular arc or elliptical.

You can use various input methods to create profile systems. You choose a method based on the type of trace curve to create.

You can use any of the following methods to create profile systems for vertical, horizontal, sloped, and curved profiles:

- Define the location line and the inclination angle from an orthogonal surface.
- Specify a distance from a reference surface or frame system.
- Copy an existing profile system to create one or multiple new profile system.
- Specify a pattern or spacing for multiple profile systems.
- Refer an existing surface and defining a non-orthogonal surface tangent to the existing surface (for example, the surface of a stiffener connected to a longitudinal frame).

NOTES

- Minimum height or distances are maintained by the software for objects that use Sketch2D, such as profiles by projection. For more information, see *Minimum Distance* (on page 92).
- When a profile system is created, a profile part is also created. Profile parts are detailed in the Structural Detailing task and then used to create manufacturing parts in the Structural Manufacturing task. Properties of the part can be modified.

- When you create a profile by projection, the dialog box title displays the plate system name using the format **Plate System:** *<name of the plate system>*.



- You can change the default colors of profile systems, seams, reference curves, and knuckle curves to enhance visibility. For more information, see *Modify Profile System Landing Curve Display Colors* (on page 783).

Stiffener Profile System by Intersection - Creates a stiffener profile system on the selected plate system by defining the landing curve of the stiffener as the intersection of the selected plate system and a grid plane. For more information, see *Stiffener Profile System by Intersection* (on page 723).

Stiffener Profile System by Offset - Creates a stiffener profile system on the selected plate system by defining the landing curve as being offset from an existing curve (a seam, plate boundary, plate edge, the landing curve of another profile system, or grid plane). For more information, see *Stiffener Profile System by Offset* (on page 739).

Stiffener Profile System by 2D Projection - Creates a stiffener profile system on the selected plate system by projecting a 2D landing curve that you have sketched onto the plate system. For more information, see *Stiffener Profile System by Projection* (on page 746).

Stiffener Profile System by Table - Creates a stiffener profile system on the selected plate system by defining the landing curve as coordinates in a table. For more information, see *Stiffener Profile System by Table* (on page 755).

Tripping Stiffener System - Creates a tripping stiffener on a selected plate system. For more information, see *Tripping Stiffener System* (on page 765).

Profile Edge Reinforcement - Places a profile system along the free edge of a plate system. For more information, see *Profile Edge Reinforcement* (on page 771).

Place Linear Member System - Places columns, beams, braces, and other linear members in the model. For more information, see *Place Linear Member Systems* (on page 787).

Place Can - Places a can in the model. A can is a reinforcing connection piece on a member to increase the surface area available for connections or to provide increased wall thickness or both. For more information, see *Place Can* (on page 860).

See Also

Knuckle Best Practices (on page 56)
Landing Curve Best Practices (on page 56)

Stiffener Profile System by Intersection

The **Stiffener Profile System by Intersection** command creates a stiffener profile system on the selected plate system by defining the landing curve of the stiffener as the intersection of the selected plate system and a grid plane.

You can control the boundaries of the stiffener profile by selecting a plate system, profile system, seam or grid plane that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the stiffener profile boundaries.

After defining the landing curve and the boundaries, you can set the profile orientation. The plate system side and the flange direction for unsymmetrical profile sections are set by the molded convention of the plate system. However, you may override the default settings by using the **Profile Properties** dialog box.

NOTE When a stiffener profile system is created, a stiffener part is also created. Profile parts are detailed in the Structural Detailing task and then used to create manufacturing parts in the Structural Manufacturing task. Properties of the part can be modified. For more information, see *Stiffener Part Properties Dialog Box* (on page 736).

Stiffener Profile System by Intersection Ribbon

Displays the options that you use to place a stiffener profile in the model.

Profile by Intersection Settings

Opens the **Profile by Intersection Settings** dialog box, which you use to view and modify the properties of the profile system that you are about to place in the model. The initial settings default from the parent plate system. For more information, see *Profile Properties Dialog Box* (on page 729).

Plate System

Specifies the plate system that you want to stiffen. You can select multiple plate systems to place multiple stiffener profiles systems at the same time. The software uses the same information for each stiffener profile system. After placement, the stiffener profiles are unrelated to each other and can be edited as individual entities.

Landing Curve Definition

Specifies the grid planes that intersect the plate system at which to place the stiffener profiles.

Profile Orientation Definition

Defines the orientation of the profile system cross-section. See the *Profile Orientation Definition Controls* section below.

Boundary Definition

Defines the boundaries of the stiffener profile. You can define the boundaries by selecting objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the stiffener profile to maintain the boundary relationship. See the *Boundary Definition Controls* section below.

Finish

Places the system using the defined parameters.

Profile Orientation Definition Controls

Available when you use the **Profile Orientation Definition** option.

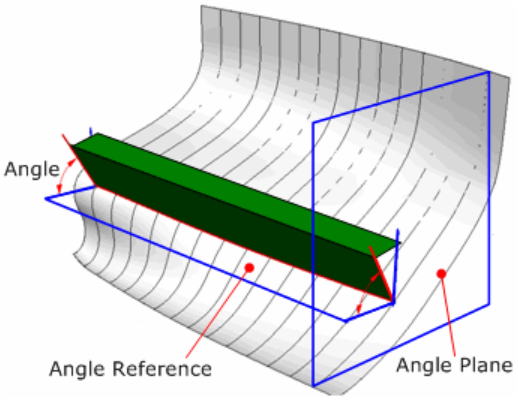
Rule Region Definition

Defines regions along the profile landing curve. Each region of the landing curve can have a different orientation rule assigned to it. The software initially defines a region boundary point at the profile boundaries. You cannot delete or modify the software-defined region boundary points. You can define and modify additional region boundary along the profile landing curve using grid/reference planes, plate design seams, and plate planning seams.

Orientation Rule Assignment

Opens a list of orientation rules from which you can select. These rules include:

- **Normal** - The web of the profile follows the plate system surface normal along the landing curve. This is the default setting if the profile parent is a plate system.
- **Up-Down** - The orientation vector of the profile cross-section remains in an up-down direction. For example, you can select **Up-Down** for profiles on a deck.
- **Fore-Aft** - The orientation vector of the profile cross-section remains in a fore-aft direction. For example, you can select **Fore-Aft** for profiles on a transverse frame.
- **Port-Starboard** - The orientation vector of the profile cross- section remains in a port-starboard direction. For example, you can select **Port-Starboard** for longitudinal stringer profiles.
- **Planar** - The orientation angle remains constant relative to a reference plane that you specify.



- **Angle** - Type the angle for the plane.
- **Reference** - Select the reference plane from which to measure the angle. You can select **Normal**, **Up-Down** (horizontal), **Fore-Aft** (longitudinal) or **Port-Starboard** (transverse). **Normal** reference plane is the plate system surface normal along the landing curve and is available for any profile orientation. **Up-Down**, **Fore-Aft** or **Port-Starboard** are available as is appropriate for the direction of the landing curve.
- **Plane** - Select the plane *in* which to measure the angle. For **Normal** Reference, this option is unavailable because the angle is always measured in a plane normal to the landing curve. For **Up-Down** or **Port-Starboard** Reference, you can select **Normal** or **Orthogonal**. If **Normal** is selected, the angle is measured in a plane normal to the landing curve. Orthogonal is the transverse, longitudinal or horizontal plane closest

to the landing curve normal. If **Orthogonal** is selected, the software selects the orthogonal plane based on average direction of the landing curve.

Twist Linear Girth

Twist is distributed evenly along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate is constant.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

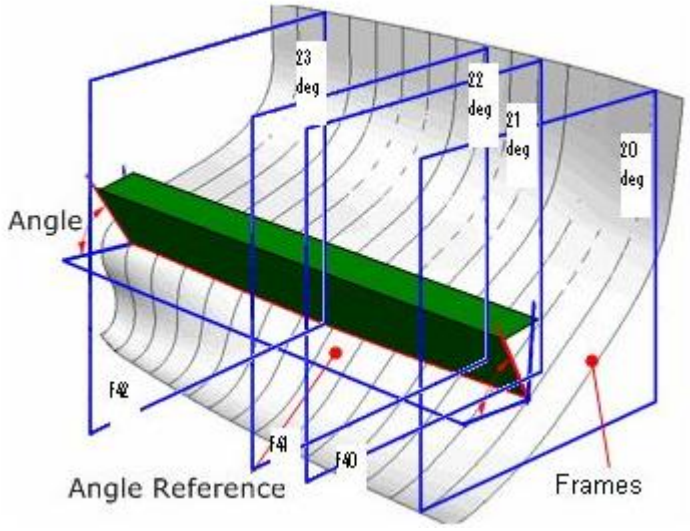
Twist Cubic

Twist is distributed using a cubic equation along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate at the ends is zero. The twist rate reaches its maximum at the midpoint.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Twist Linear Ortho

Twist is constant at every frame along the orthogonal projection length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The software uses the Main_CS frame system if it exists. Otherwise, the software uses the CS_0 frame system.



- **First Angle** - Type the angle at the first (beginning) point of the region boundary.

- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Profile System Table

Opens a read-only table containing the coordinates, orientation angles and twist rates of the current profile.

Boundary Definition Controls

Available when you use the **Boundary Definition** option.

Pick Boundaries

Selects objects in the model or in the Workspace Explorer to define the boundaries. This option is only available when you are defining the boundaries.

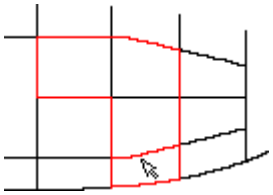
TIP After you select a boundary plate, press SHIFT to select the **Pick Edges** option. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Selects free edges on plates to define the boundary. This option is available when you select a plate in the **Pick Boundaries** step.

Solve Ambiguity

Clarifies the required boundary. If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to specify the boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Draws the boundaries. This option is only available when you are defining the boundaries. See *Define Plane Controls* below.

Sketching Plane

Selects the sketching plane for the boundary.

Add Intersecting Item

Allows you to select objects in the model that intersect the sketching plane to use as reference geometry. You see the selected objects in the Sketch 2D environment.

 **Add Projection Item**

Allows you to select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment.

 **Sketch 2D**

Opens the Sketch 2D environment in which you can draw the boundary.

Auto

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. The Workspace Explorer and the graphics view highlight items in the select set. If the software adds no objects to the select set, a message displays in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

 **Reject**

Clears all selected objects.

 **Accept**

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Section Type

Specifies the profile cross-section type.

Section Size

Specifies the profile section size.

Primary Orientation

Defines the side of the plate system used to place the stiffener profile. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Secondary Orientation

Defines the direction of the web thickness for symmetrical cross-sections and the direction of the flange and web thickness for unsymmetrical cross- sections. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Start Connection

Defines the method of connecting the beginning of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

End Connection

Defines the method of connecting the end of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

What do you want to do?

- *Place stiffener profile by intersection* (on page 728)
- *Modify a stiffener profile* (on page 729)
- *Delete a stiffener profile* (on page 729)

Place stiffener profile by intersection

1. Click **Stiffener Profile System by Intersection** on the vertical toolbar.
2. Select the plate system to stiffen.
3. Select the **Section Type** and **Section Size**.
4. Click **Landing Curve Definition**.
5. Select a grid plane that intersects the plate system that you selected.
6. Click **Profile Orientation Definition**.
7. Define the orientation of the stiffener profile with regard to the plate system.

8. Click **Boundary Definition** .
9. Define the boundaries for the stiffener profile.
10. Click **Accept** .
11. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
12. Click **Finish**.

NOTES

- Stiffener profiles are always associated with the plate system to which they belong.
- You can select more than one grid plane to place multiple stiffener profiles at one time.

Modify a stiffener profile

1. Click **Select** on the vertical toolbar.
2. Select **Root Profiles** in the **Locate Filter**.
3. Select the stiffener profile to modify.
4. Using the ribbon controls, modify the stiffener profile as needed.

Delete a stiffener profile

1. Click **Select** on the vertical toolbar.
2. Select **Root Profiles** in the **Locate Filter**.
3. Select the stiffener profile to delete.
4. Click **Delete** .

Profile Properties Dialog Box

Specifies the properties for the root or leaf profile system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.

- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Profile Properties Dialog Box) (on page 730)

Cross Section Tab (Profile Properties Dialog Box) (on page 732)

Section Orientation Tab (Profile Properties Dialog Box) (on page 732)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Main Tab (Profile Properties Dialog Box)

Specifies the general settings for the profile.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Specifies how the name is created.

StdProfileSystemNamingRule - Uses the following syntax: <Parent Name>-<Index Number><Profile Type>-<Index Number>. For example, L3-1LBH-1-1L-1, where L3-1LBH-1 is the parent name, 1L is an index number appended by the profile type, and 1 is an index number. The profile type abbreviations are L for longitudinal, V for vertical, T for Transversal, R for ring, and A for axial.

User Defined - Indicates that you want to name the profile yourself by using the appropriate box.

Type

Specifies the type of profile. The profile type is used by the naming rule in naming the profile system and by the molded conventions assigned to the parent plate system.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Displays the parent system for the object.

Landing Curve Definition Method

Displays the name of the profile system command used to create the profile.

Specification

Specifies the specification for the object. This property is inherited from the parent system.

Description

Specifies a description for the object.

Continuity

Defines the default behavior for the object when that object intersects another plate system or profile. To change the options on the list, edit the **Slab Continuity** select list in Catalog.

Continuous indicates that this object penetrates other plate systems and profile systems that the object intersects.

Intercostal indicates that this object is split by the other plate systems and profile systems that the object intersects.

This property can only be modified at the root system.

Active

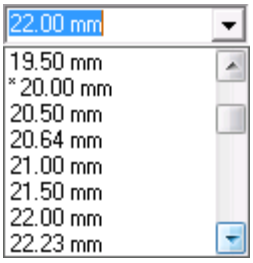
Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Continuity Priority

Specifies the priority when two profile systems that have the same **Continuity** setting intersect. The profile system with the lowest continuity priority number penetrates the other system. For example, two profile systems intersect and the **Continuity** setting of each set to **Continuous**. The profile system with the lowest **Continuity Priority** number splits the other profile system. You can only modify this property at the root system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Cross Section Tab (Profile Properties Dialog Box)

Specifies the cross-section settings for the profile.

Section Type

Specifies the profile cross-section type. The software displays the section type properties for the section type that you selected. Each section type has different properties.

Section Size

Specifies the profile section size.

Material

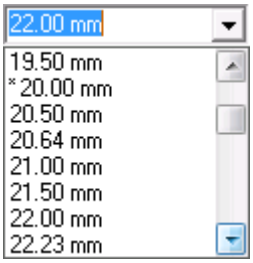
Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Section Orientation Tab (Profile Properties Dialog Box)

Specifies the position of the profile cross-section.

Primary Orientation

Defines the side of the plate system used to place the stiffener profile. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Secondary Orientation

Defines the direction of the web thickness for symmetrical cross-sections and the direction of the flange and web thickness for unsymmetrical cross- sections. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Mounting Face

Specifies the cross-section face to mount to the plate system. The options vary depending on the profile cross-section.

Load Point

Specifies the cross-section load point through which the landing curve passes.

Landing Curve Projection Method

Specifies how the software locates the landing curve with respect to the parent plate system thickness. The list is defined by the ShipStructLandingCurveProjectionMethod codelist

Along Web indicates that Smart 3D projects the landing curve in-line with the profile web.



Normal indicates that Smart 3D projects the landing curve normal to the plate surface.



Attachment Method

Specifies the method of trimming the profile web where it is attached to the plate system. The list is defined by the ShipStructStiffenerAttachmentMethod codelist.

Partial Trim indicates that Smart 3D trims the short side of the web.



No Trim indicates that Smart 3D does not trim the web.



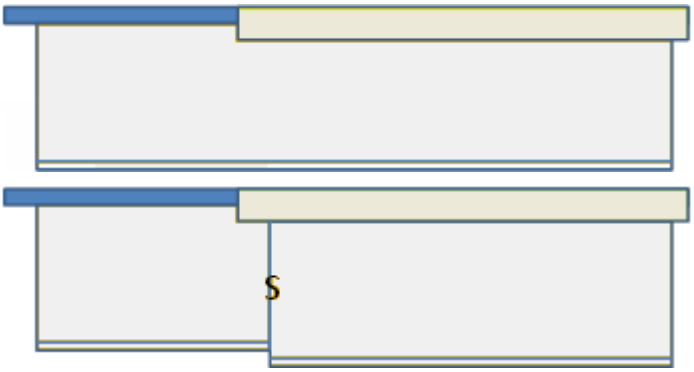
Full Trim indicates that Smart 3D trims the entire web.



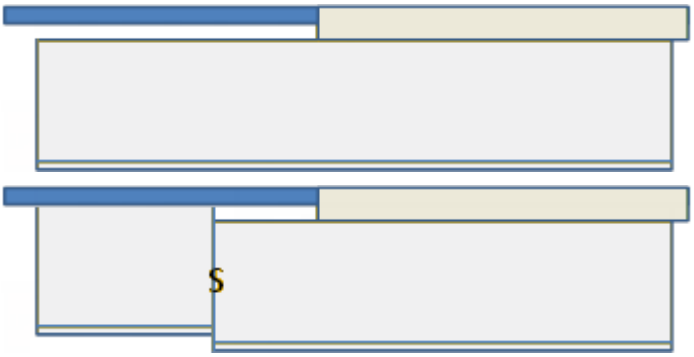
Offset Method

Specifies how the profile is offset from and trimmed by the plate system when the profile crosses design seams with leaf plates of varying thickness, and the profile is on the thickened sides of the plates. The list is defined by the StiffenerOffsetMethod codelist.

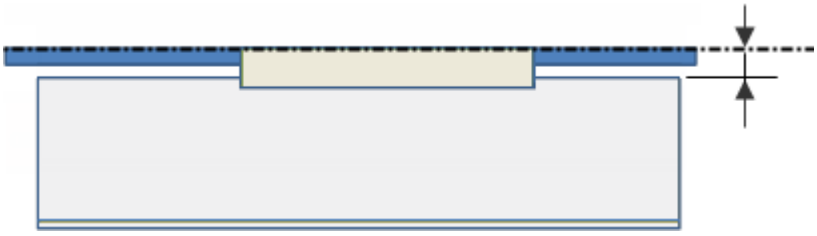
Position on thinnest plate indicates that each profile part of the parent profile system is mounted to the thinnest plate part that the profile part crosses. The profile part is then trimmed to any thicker plate it crosses.



Position on thickest plate indicates that each profile part of the parent profile system is mounted to the thickest plate part that the profile part crosses. No trimming occurs.



Offset and trim indicates that each profile part of the parent profile system is mounted at the specified **Offset Value** from the molded surface of the plate system. Profile part position is not adjusted based on plate part thickness. Each profile part is trimmed by any plate part thicker than the offset distance.



Offset with no trim indicates that each profile part of the parent profile system is mounted at the specified **Offset Value** from the molded surface of the plate system. Profile part position is not adjusted based on plate part thickness. Profile parts are not trimmed by any plate part thicker than the offset distance.



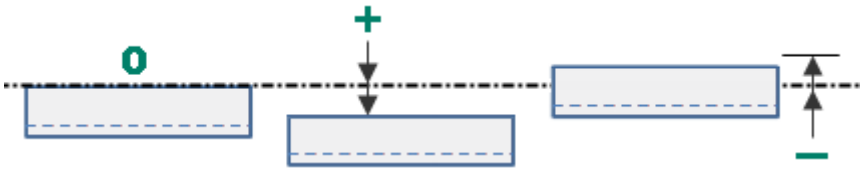
This option is useful when the gap from successively thicker plates can be filled by welding:



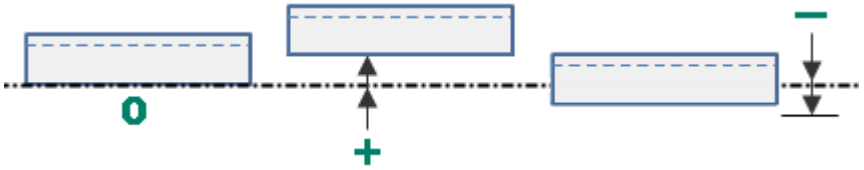
Offset Value

Defines the amount of offset used with **Offset and trim** or **Offset with no trim**. A positive value moves the stiffener away from the molded surface. A negative value moves it toward the molded surface. The value is also relative to the profile orientation.

For example, the following positive and negative values are used when profile orientation is **Below** on a deck:



The following positive and negative values are used when the profile orientation is **Above** on a deck:



NOTE Attachment, connection, and offset methods are applied to the profile after the plate and profile systems are detailed in Structural Detailing.

Start Connection Method

Defines the method of connecting the beginning of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

End Connection Method

Defines the method of connecting the end of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

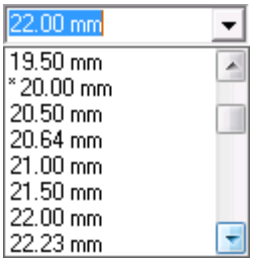
Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Stiffener Part Properties Dialog Box

Specifies the properties for the stiffener part that you are editing. Default properties are inherited from the parent system of the part.

Main Tab (Stiffener Part Properties Dialog Box) (on page 737)

Profile Section Tab (Stiffener Part Properties Dialog Box) (on page 738)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (Stiffener Part Properties Dialog Box) (on page 738)

Weight & CG Tab (on page 129)

Extended User Attributes Tab (on page 129)

Routing Tab (on page 129)

Main Tab (Stiffener Part Properties Dialog Box)

Specifies the general properties of the stiffener part.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Specifies the naming rule to use to name the stiffener part.

- Select **StiffenerPartRule** to create a name based on the parent root profile system name, the location within the parent root profile system, and the parent assembly name.
- Select **User Defined** to name the stiffener part using the appropriate box.

Type

Displays the type of the part. It is inherited from the parent system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the category for the stiffener part. Categories specify the role of the part in the model.

Parent System

Displays the parent system for the stiffener part.

Landing Curve Definition Method

Displays the landing curve definition.

Specification

Displays the structural specification for the stiffener part. It is inherited from the parent system.

Description

Type a description for the stiffener part.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Profile Section Tab (Stiffener Part Properties Dialog Box)

Specifies the profile section settings for the stiffener part.

Material

Displays the material for the stiffener part.

Grade

Displays the material grade for the stiffener part.

Section Type

Displays the profile section type.

Section Size

Displays the profile section size.

General Tab (Stiffener Part Properties Dialog Box)

Specifies general properties of the stiffener part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Category: Standard

ProfileType

Specifies the profile type of the part. The list is defined by the StrMfgProfileType codelist.

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Stiffener Part Properties Dialog Box)* (on page 737).

Category: Planning

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

Stiffener Profile System by Offset

The **Stiffener Profile System by Offset** command creates a stiffener profile system on the selected plate system by defining the landing curve as being offset from an existing curve (a seam, plate boundary, plate edge, the landing curve of another profile system, or grid plane).

You can control the boundaries of the stiffener profile by selecting a plate system, profile system, seam or grid plane that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the stiffener profile boundaries.

After defining the landing curve and the boundaries, you can set the profile orientation. The plate system side and the flange direction for unsymmetrical profile sections are set by the molded convention of the plate system. However, you may override the default settings by using the **Profile Properties** dialog box.

NOTE When a stiffener profile system is created, a stiffener part is also created. Profile parts are detailed in the Structural Detailing task and then used to create manufacturing parts in the Structural Manufacturing task. Properties of the part can be modified. For more information, see *Stiffener Part Properties Dialog Box* (on page 736).

Stiffener Profile System by Offset Ribbon

Displays the options that you use to place a stiffener profile in the model.

Profile by Offset Settings

Opens the **Profile by Offset Settings** dialog box, which you use to view and modify the properties of the profile system that you are about to place in the model. The initial settings default from the parent plate system. For more information, see *Profile Properties Dialog Box* (on page 729).

Plate System

Specifies the plate system that you want to stiffen. You can select multiple plate systems to place multiple stiffener profiles systems at the same time. The software uses the same information for each stiffener profile system. After placement, the stiffener profiles are unrelated to each other and can be edited as individual entities.

Base Curve Definition

Specifies an existing curve or grid plane from which to offset the profile. The intersection of the grid plane and the plate system defines the base curve.

Offset Curve Definition

Defines the offset parameters and direction from the base curve. See the *Offset Curve Definition Controls* section below.

Profile Orientation Definition

Defines the orientation of the profile system cross-section. See the *Profile Orientation Definition Controls* section below.

Boundary Definition

Defines the boundaries of the stiffener profile. You can define the boundaries by selecting objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the stiffener profile to maintain the boundary relationship. See

the *Boundary Definition Controls* section below.

Finish

Places the system using the defined parameters.

Offset Curve Definition Controls

Available when you use the **Offset Curve Definition** option.

Unlimited

Defines offset curves without a stop limit.

Stop Limit

Defines offset curves with a stop limit. By using a stop limit, you can select an edge, profile landing curve, connection reference curve, or seam to control the landing curves being created. If you define an offset value and the **Offset lock** is locked, then landing curves are applied at increments of the offset value until the selected stop limit is reached. If the **Offset lock** is unlocked and a value has been defined for **Count**, then the landing curves equal to the number entered in the **Count** box are equally spaced between the base curve and the stop limit.

Offset lock

Locks the **Offset** value, disabling updates of the offset value by mouse moves.

Offset

Specifies the offset distance between the base curve and the landing curve that you are defining. If you type **0,** the landing curve is placed on the base curve. This box may also display the offset distance as defined by mouse movements.

Method

Select an offset direction. You can select transverse, longitudinal, vertical, or girth.

Count

Specifies how many profiles to create, each equally spaced by increments of the offset value.

Step

Defines an increment that the system uses as you dynamically move the mouse. If you have entered a value in the **Offset** box, then this control is not used.

Profile Orientation Definition Controls

Available when you use the **Profile Orientation Definition** option.

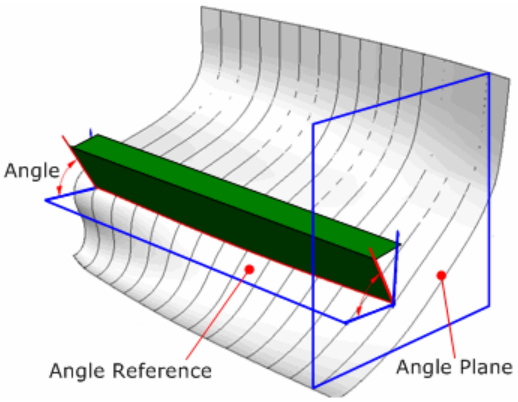
Rule Region Definition

Defines regions along the profile landing curve. Each region of the landing curve can have a different orientation rule assigned to it. The software initially defines a region boundary point at the profile boundaries. You cannot delete or modify the software-defined region boundary points. You can define and modify additional region boundary along the profile landing curve using grid/reference planes, plate design seams, and plate planning seams.

Orientation Rule Assignment

Opens a list of orientation rules from which you can select. These rules include:

- **Normal** - The web of the profile follows the plate system surface normal along the landing curve. This is the default setting if the profile parent is a plate system.
- **Up-Down** - The orientation vector of the profile cross-section remains in an up-down direction. For example, you can select **Up-Down** for profiles on a deck.
- **Fore-Aft** - The orientation vector of the profile cross-section remains in a fore-aft direction. For example, you can select **Fore-Aft** for profiles on a transverse frame.
- **Port-Starboard** - The orientation vector of the profile cross- section remains in a port-starboard direction. For example, you can select **Port-Starboard** for longitudinal stringer profiles.
- **Planar** - The orientation angle remains constant relative to a reference plane that you specify.



- **Angle** - Type the angle for the plane.
- **Reference** - Select the reference plane from which to measure the angle. You can select **Normal**, **Up-Down** (horizontal), **Fore-Aft** (longitudinal) or **Port-Starboard** (transverse). **Normal** reference plane is the plate system surface normal along the landing curve and is available for any profile orientation. **Up-Down**, **Fore-Aft** or **Port-Starboard** are available as is appropriate for the direction of the landing curve.
- **Plane** - Select the plane *in* which to measure the angle. For **Normal** Reference, this option is unavailable because the angle is always measured in a plane normal to the landing curve. For **Up-Down** or **Port-Starboard** Reference, you can select **Normal** or **Orthogonal**. If **Normal** is selected, the angle is measured in a plane normal to the landing curve. Orthogonal is the transverse, longitudinal or horizontal plane closest to the landing curve normal. If **Orthogonal** is selected, the software selects the orthogonal plane based on average direction of the landing curve.

Twist Linear Girth

Twist is distributed evenly along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate is constant.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.

- **Plane** - See the definition and graphic for the **Planar** orientation rule.

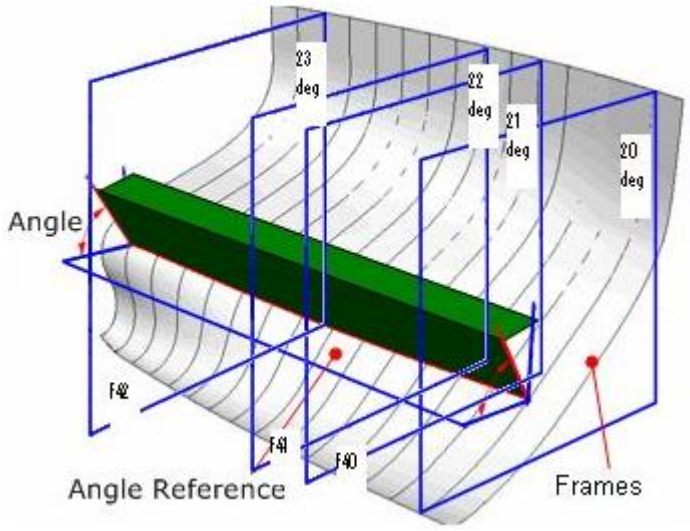
Twist Cubic

Twist is distributed using a cubic equation along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate at the ends is zero. The twist rate reaches its maximum at the midpoint.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Twist Linear Ortho

Twist is constant at every frame along the orthogonal projection length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The software uses the Main_CS frame system if it exists. Otherwise, the software uses the CS_0 frame system.



- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Profile System Table

Opens a read-only table containing the coordinates, orientation angles and twist rates of the current profile.

Boundary Definition Controls

Available when you use the **Boundary Definition** option.

Pick Boundaries

Selects objects in the model or in the Workspace Explorer to define the boundaries. This option is only available when you are defining the boundaries.

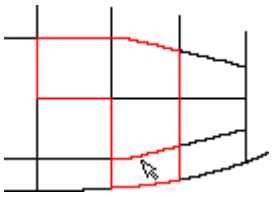
TIP After you select a boundary plate, press SHIFT to select the **Pick Edges** option. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Selects free edges on plates to define the boundary. This option is available when you select a plate in the **Pick Boundaries** step.

Solve Ambiguity

Clarifies the required boundary. If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to specify the boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Draws the boundaries. This option is only available when you are defining the boundaries. See *Define Plane Controls* below.

Sketching Plane

Selects the sketching plane for the boundary.

Add Intersecting Item

Allows you to select objects in the model that intersect the sketching plane to use as reference geometry. You see the selected objects in the Sketch 2D environment.

Add Projection Item

Allows you to select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the boundary.

Auto

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. The Workspace Explorer and the graphics view highlight items in the select set. If the software adds no objects to the select set, a message displays in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Section Type

Specifies the profile cross-section type.

Section Size

Specifies the profile section size.

Primary Orientation

Defines the side of the plate system used to place the stiffener profile. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Secondary Orientation

Defines the direction of the web thickness for symmetrical cross-sections and the direction of the flange and web thickness for unsymmetrical cross- sections. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Start Connection

Defines the method of connecting the beginning of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

End Connection

Defines the method of connecting the end of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

What do you want to do?

- *Place stiffener profile by offset* (on page 745)
- *Modify a stiffener profile* (on page 729)
- *Delete a stiffener profile* (on page 729)

Place stiffener profile by offset

1. Click **Stiffener Profile System by Offset** on the vertical toolbar.
2. Select the plate system to stiffen.
3. Select the **Section Type** and **Section Size**.
4. Click **Base Curve Definition**.
5. Select a base curve (grid plane) that intersects the plate system that you selected. This is the curve to offset from.
6. Click **Offset Curve Definition**.
7. Define the offset from the base curve.
8. Click **Profile Orientation Definition**.
9. Define the orientation of the profile system cross-section.
10. Click **Boundary Definition**.
11. Define the boundaries for the stiffener profile.
12. Click **Accept**.
13. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

Solve ambiguous solution created by selected boundaries (on page 82)

14. Click **Finish**.

NOTE Stiffener profiles are always associated with the plate system to which they belong.

Stiffener Profile System by Projection


The **Stiffener Profile System by Projection** command creates a stiffener profile system on the selected plate system by projecting a 2D or 3D landing curve that you have sketched onto the plate system.

You can control the boundaries of the stiffener profile by selecting a plate system, profile system, seam or grid plane that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the stiffener profile boundaries.

After defining the landing curve and the boundaries, you can set the profile orientation. The plate system side and the flange direction for unsymmetrical profile sections are set by the molded convention of the plate system. However, you may override the default settings by using the **Profile Properties** dialog box.

NOTE When a stiffener profile system is created, a stiffener part is also created. Profile parts are detailed in the Structural Detailing task and then used to create manufacturing parts in the Structural Manufacturing task. Properties of the part can be modified. For more information, see *Stiffener Part Properties Dialog Box* (on page 736).

Profile Knuckles

If the plate system associated with the profile has an angle with no radius on the extrusion curve, the software creates a profile knuckle object. You can select this object graphically, or in the **Workspace Explorer**. For more information, see *Profile Knuckle Properties Dialog Box* (on page 933).

Stiffener Profile System by Projection Ribbon

Displays the options that you use to place a stiffener profile in the model.


Profile by Intersection Settings

Opens the **Profile by Intersection Settings** dialog box, which you use to view and modify the properties of the profile system that you are about to place in the model. The initial settings default from the parent plate system. For more information, see *Profile Properties Dialog Box* (on page 729).

Plate System

Specifies the plate system that you want to stiffen. You can select multiple plate systems to place multiple stiffener profiles systems at the same time. The software uses the same information for each stiffener profile system. After placement, the stiffener profiles are unrelated to each other and can be edited as individual entities.


Define using 2D Sketcher

Defines the landing curve by drawing it in the Sketch 2D environment. See the *Landing Curve Definition Controls* and *Define Plane Controls* sections below.


Landing Curve Definition by 3D

Allows creation of the landing curve directly in a graphic view by using the **Profile Palette** dialog box (similar to the **Geometric Construction Palette**), the **Geometric Construction** ribbon, and the **Geometric Construction Explorer**. This option is the 3D equivalent to Sketch 2D. For more information, see *Geometric Constructions* (on page 1027).

Profile Orientation Definition

Defines the orientation of the profile system cross-section. See the *Profile Orientation Definition Controls* section below.

Boundary Definition

Defines the boundaries of the stiffener profile. You can define the boundaries by selecting objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the stiffener profile to maintain the boundary relationship. See the *Boundary Definition Controls* section below.

Finish

Places the system using the defined parameters.

Profile Orientation Definition Controls

Available when you use the **Profile Orientation Definition** option.

Rule Region Definition

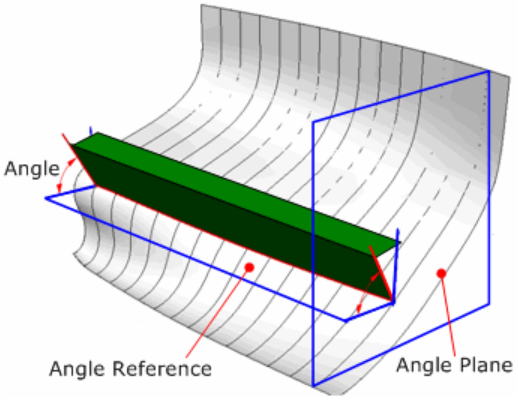
Defines regions along the profile landing curve. Each region of the landing curve can have a different orientation rule assigned to it. The software initially defines a region boundary point at the profile boundaries. You cannot delete or modify the software-defined region boundary points. You can define and modify additional region boundary along the profile landing curve using grid/reference planes, plate design seams, and plate planning seams.

Orientation Rule Assignment

Opens a list of orientation rules from which you can select. These rules include:

- **Normal** - The web of the profile follows the plate system surface normal along the landing curve. This is the default setting if the profile parent is a plate system.
- **Up-Down** - The orientation vector of the profile cross-section remains in an up-down direction. For example, you can select **Up-Down** for profiles on a deck.
- **Fore-Aft** - The orientation vector of the profile cross-section remains in a fore-aft direction. For example, you can select **Fore-Aft** for profiles on a transverse frame.
- **Port-Starboard** - The orientation vector of the profile cross- section remains in a port-starboard direction. For example, you can select **Port-Starboard** for longitudinal stringer profiles.

- **Planar** - The orientation angle remains constant relative to a reference plane that you specify.



- **Angle** - Type the angle for the plane.
- **Reference** - Select the reference plane from which to measure the angle. You can select **Normal**, **Up-Down** (horizontal), **Fore-Aft** (longitudinal) or **Port-Starboard** (transverse). **Normal** reference plane is the plate system surface normal along the landing curve and is available for any profile orientation. **Up-Down**, **Fore-Aft** or **Port-Starboard** are available as is appropriate for the direction of the landing curve.
- **Plane** - Select the plane *in* which to measure the angle. For **Normal** Reference, this option is unavailable because the angle is always measured in a plane normal to the landing curve. For **Up-Down** or **Port-Starboard** Reference, you can select **Normal** or **Orthogonal**. If **Normal** is selected, the angle is measured in a plane normal to the landing curve. Orthogonal is the transverse, longitudinal or horizontal plane closest to the landing curve normal. If **Orthogonal** is selected, the software selects the orthogonal plane based on average direction of the landing curve.

Twist Linear Girth

Twist is distributed evenly along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate is constant.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Twist Cubic

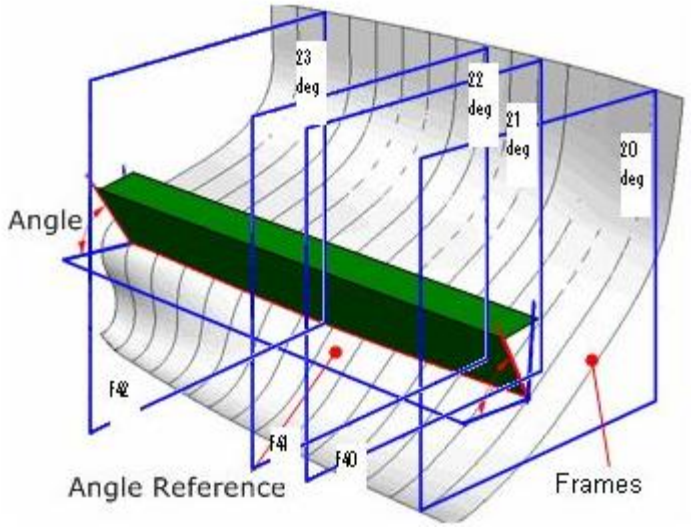
Twist is distributed using a cubic equation along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate at the ends is zero. The twist rate reaches its maximum at the midpoint.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.

- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Twist Linear Ortho

Twist is constant at every frame along the orthogonal projection length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The software uses the Main_CS frame system if it exists. Otherwise, the software uses the CS_0 frame system.



- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Profile System Table

Opens a read-only table containing the coordinates, orientation angles and twist rates of the current profile.

Boundary Definition Controls

Available when you use the **Boundary Definition** option.

Pick Boundaries

Selects objects in the model or in the Workspace Explorer to define the boundaries. This option is only available when you are defining the boundaries.

TIP After you select a boundary plate, press SHIFT to select the **Pick Edges** option. Press SHIFT again to return to the **Pick Boundaries** option.

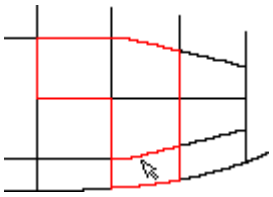
Pick Edges

Selects free edges on plates to define the boundary. This option is available when you select

a plate in the **Pick Boundaries** step.

Solve Ambiguity

Clarifies the required boundary. If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to specify the boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).



Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Boundary Curve

Draws the boundaries. This option is only available when you are defining the boundaries. See *Define Plane Controls* below.

Sketching Plane

Selects the sketching plane for the boundary.

Add Intersecting Item

Allows you to select objects in the model that intersect the sketching plane to use as reference geometry. You see the selected objects in the Sketch 2D environment.

Add Projection Item

Allows you to select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the boundary.

Auto

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. The Workspace Explorer and the graphics view highlight items in the select set. If the software adds no objects to the select set, a message displays in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

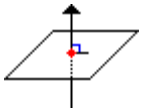
Define Plane Controls

Available when you use the **Define using 2D Sketcher** option.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only

available when you use the **Plane by Point and Vector** plane method.

 **Define Point 1**

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

 **Define Point 2**

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

 **Define Point 3**

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

 **Reject**

Clears all selected objects.

 **Accept**

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Section Type

Specifies the profile cross-section type.

Section Size

Specifies the profile section size.

Primary Orientation

Defines the side of the plate system used to place the stiffener profile. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Secondary Orientation

Defines the direction of the web thickness for symmetrical cross-sections and the direction of the flange and web thickness for unsymmetrical cross- sections. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Start Connection

Defines the method of connecting the beginning of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

End Connection

Defines the method of connecting the end of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

Knuckle Best Practices **(on page 56)**

What do you want to do?

- *Place stiffener profile by projection* (on page 753)
- *Define landing curve by 2D* (on page 87)
- *Define landing curve by 3D* (on page 88)
- *Modify a stiffener profile* (on page 729)
- *Delete a stiffener profile* (on page 729)

Place stiffener profile by projection

1. Click **Stiffener Profile System by Projection** on the vertical toolbar.
2. Select the plate system to stiffen.
3. Select the **Section Type** and **Section Size**.
4. Define the landing curve using one of the following methods:

   Define landing curve by 2D (on page 87) .

Define landing curve by 3D (on page 88) .

5. Click **Profile Orientation Definition** .
6. Define the orientation of the stiffener profile with regard to the plate system.
7. Click **Boundary Definition** .
8. Define the boundaries for the stiffener profile.
9. Click **Accept** .
10. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
11. Click **Finish**.

NOTES

- If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting profile curves.
- Stiffener profiles are always associated with the plate system to which they belong.
- If the plate system is symmetrical about the sketch plane (For example, a revolved plate system or a hull), an unbounded sketched seam curve appear on both sides of the plate system. If you want the profile on only one side, then you must add a boundary to the profile so that the command switches to **Solve Ambiguity** . The plate system on which the profile is placed can also be selected as its own boundary to meet this requirement.

Stiffener Profile System by Table

The **Stiffener Profile System by Table** command creates a stiffener profile system on the selected plate system by defining the landing curve as coordinates in a table. This command is particularly useful when placing stiffener profiles on a hull or other multi-curved shape.

You can control the boundaries of the stiffener profile by selecting a plate system, profile system, seam or grid plane that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the stiffener profile boundaries.

After defining the landing curve and the boundaries, you can set the profile orientation. The plate system side and the flange direction for unsymmetrical profile sections are set by the molded convention of the plate system. However, you may override the default settings by using the **Profile Properties** dialog box.

NOTE When a stiffener profile system is created, a stiffener part is also created. Profile parts are detailed in the Structural Detailing task and then used to create manufacturing parts in the Structural Manufacturing task. Properties of the part can be modified. For more information, see *Stiffener Part Properties Dialog Box* (on page 736).

Stiffener Profile System by Table Ribbon

Displays the options that you use to place a stiffener profile in the model.

Profile Properties

Opens the **Profile Properties** dialog box, which you use to view and modify the properties of the profile system that you are about to place in the model. The initial settings default from the parent plate system. For more information, see *Profile Properties Dialog Box* (on page 729).

Plate System

Specifies the plate system that you want to stiffen. You can select multiple plate systems to place multiple stiffener profiles systems at the same time. The software uses the same information for each stiffener profile system. After placement, the stiffener profiles are unrelated to each other and can be edited as individual entities.

Landing Curve Table

Opens a table where you can define the landing curve coordinates for the stiffener profile system. For more information, see *Landing Curve Table Dialog Box* (on page 760).

Profile Orientation Definition

Defines the orientation of the profile system cross-section. See the *Profile Orientation Definition Controls* section below.

Boundary Definition

Defines the boundaries of the stiffener profile. You can define the boundaries by selecting objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the stiffener profile to maintain the boundary relationship. See the *Boundary Definition Controls* section below.

Finish

Places the system using the defined parameters.

Profile Orientation Definition Controls

Available when you use the **Profile Orientation Definition** option.

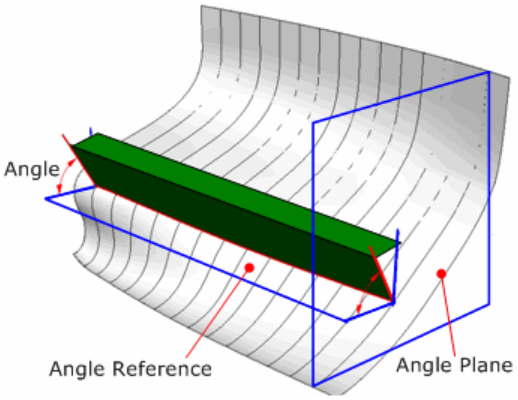
Rule Region Definition

Defines regions along the profile landing curve. Each region of the landing curve can have a different orientation rule assigned to it. The software initially defines a region boundary point at the profile boundaries. You cannot delete or modify the software-defined region boundary points. You can define and modify additional region boundary along the profile landing curve using grid/reference planes, plate design seams, and plate planning seams.

Orientation Rule Assignment

Opens a list of orientation rules from which you can select. These rules include:

- **Normal** - The web of the profile follows the plate system surface normal along the landing curve. This is the default setting if the profile parent is a plate system.
- **Up-Down** - The orientation vector of the profile cross-section remains in an up-down direction. For example, you can select **Up-Down** for profiles on a deck.
- **Fore-Aft** - The orientation vector of the profile cross-section remains in a fore-aft direction. For example, you can select **Fore-Aft** for profiles on a transverse frame.
- **Port-Starboard** - The orientation vector of the profile cross- section remains in a port-starboard direction. For example, you can select **Port-Starboard** for longitudinal stringer profiles.
- **Planar** - The orientation angle remains constant relative to a reference plane that you specify.



- **Angle** - Type the angle for the plane.
- **Reference** - Select the reference plane from which to measure the angle. You can select **Normal**, **Up-Down** (horizontal), **Fore-Aft** (longitudinal) or **Port-Starboard** (transverse). **Normal** reference plane is the plate system surface normal along the landing curve and is available for any profile orientation. **Up-Down**, **Fore-Aft** or **Port-Starboard** are available as is appropriate for the direction of the landing curve.
- **Plane** - Select the plane *in* which to measure the angle. For **Normal** Reference, this option is unavailable because the angle is always measured in a plane normal to the landing curve. For **Up-Down** or **Port-Starboard** Reference, you can select **Normal** or **Orthogonal**. If **Normal** is selected, the angle is measured in a plane normal to the landing curve. Orthogonal is the transverse, longitudinal or horizontal plane closest

to the landing curve normal. If **Orthogonal** is selected, the software selects the orthogonal plane based on average direction of the landing curve.

Twist Linear Girth

Twist is distributed evenly along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate is constant.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

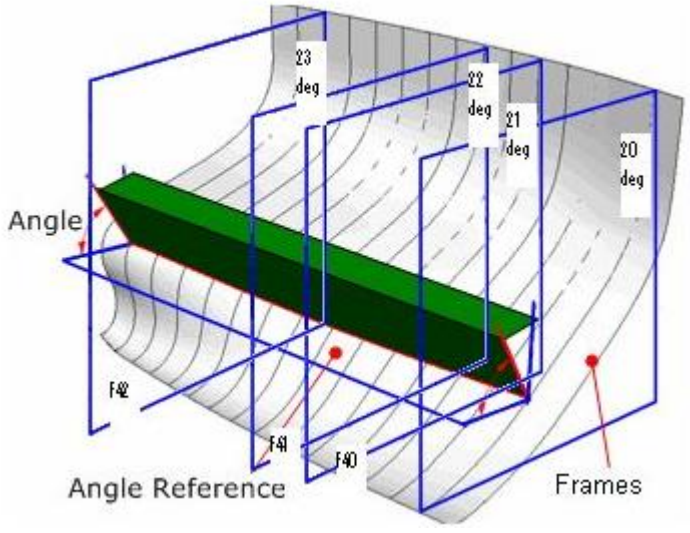
Twist Cubic

Twist is distributed using a cubic equation along the girth length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The twist rate at the ends is zero. The twist rate reaches its maximum at the midpoint.

- **First Angle** - Type the angle at the first (beginning) point of the region boundary.
- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Twist Linear Ortho

Twist is constant at every frame along the orthogonal projection length of the profile region, based on the difference between the **First Angle** and **Second Angle** defined at each end of the region. The software uses the Main_CS frame system if it exists. Otherwise, the software uses the CS_0 frame system.



- **First Angle** - Type the angle at the first (beginning) point of the region boundary.

- **Second Angle** - Type the angle at the second (end) point of the region boundary.
- **Angle** - See the definition and graphic for the **Planar** orientation rule.
- **Reference** - See the definition and graphic for the **Planar** orientation rule.
- **Plane** - See the definition and graphic for the **Planar** orientation rule.

Profile System Table

Opens a read-only table containing the coordinates, orientation angles and twist rates of the current profile.

Boundary Definition Controls

Available when you use the **Boundary Definition** option.

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Section Type

Specifies the profile cross-section type.

Section Size

Specifies the profile section size.

Primary Orientation

Defines the side of the plate system used to place the stiffener profile. The default setting is determined by the molded conventions of the parent system. This property can be modified

only at the root system.

Secondary Orientation

Defines the direction of the web thickness for symmetrical cross-sections and the direction of the flange and web thickness for unsymmetrical cross- sections. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Start Connection

Defines the method of connecting the beginning of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

End Connection

Defines the method of connecting the end of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

Landing Curve Table Dialog Box **(on page 760)**

What do you want to do?

- *Place stiffener profile by table* (on page 760)
- *Modify a stiffener profile* (on page 729)
- *Delete a stiffener profile* (on page 729)

Place stiffener profile by table

1. Click **Stiffener Profile System by Table** on the vertical toolbar.
2. Select the plate system to stiffen.
3. Select the **Section Type** and **Section Size**.
4. Click **Landing Curve Table**.
5. Define the landing curve coordinates in the table. For more information about defining coordinates, see *Landing Curve Table Dialog Box* (on page 760).
6. Click **Close**.
7. Click **Profile Orientation Definition**.
8. Define the orientation of the stiffener profile with regard to the plate system.
9. Click **Boundary Definition**.
10. Define the boundaries for the stiffener profile.
11. Click **Accept**.
12. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
13. Click **Finish**.

NOTES

- If you do not define boundaries, the software uses the ends of the landing curve as the boundaries.
- Stiffener profiles are always associated with the plate system to which they belong.

Landing Curve Table Dialog Box

Specifies a landing curve by defining X, Y, and Z coordinates of points along the landing curve. Each point along the landing curve is defined on a single row in the table. You must define two out of the three point coordinates. The software derives the third coordinate for you, and highlights the derived coordinate in blue.

Dialog Box Controls

	Point Type	Intersecting Object	X		Y		Z	
			Reference	Offset	Reference	Offset	Reference	**Offset**
1	Definition		F36	500.00 mm	15861.24 mm	0.00 mm	10325.00 mm	0.00 mm
2	Knuckle		F30	0.00 mm	15541.31 mm	0.00 mm	11000.00 mm	0.00 mm
3	Definition	F28-1TBH	28000.00 mm	0.00 mm	15296.79 mm	0.00 mm	10723.00 mm	0.00 mm
4	Definition		F24	0.00 mm	14614.09 mm	0.00 mm	10445.00 mm	0.00 mm

Point Type

Select an option for each point:

- **Definition** - The point on the row is used to define the landing curve, and the curve transitions smoothly through the point.
- **Knuckle** - The point on the row is used to define the landing curve, and the curve knuckles at the point.

Display - The point is computed based on the intersection of the landing curve and another object, as defined by **Derived Points in the Table**. Display points are not definition points and are not saved as part of the landing curve definition.

Intersecting Object - Select a seam or profile stiffener on the selected plate system, or another plate system that is bounded or split by the selected plate system.

- If **Intersecting Object** is not selected, then two of the X, Y, and Z coordinates must have values. The third coordinate is derived.
- If **Intersecting Object** is selected, then one of the X, Y, and Z coordinates must have a value. The other two coordinates are derived.

X Reference - Select a frame (Y-Z) grid plane to specify the X-coordinate of the landing curve point. You can also explicitly specify the X coordinate by typing the point's location along the X-axis.

X Offset - Type an offset value from the X reference.

Y Reference - Select a buttock (X-Z) grid plane to specify the Y-coordinate of the landing curve point. You can also explicitly specify the Y coordinate by typing the point's location along the Y-axis.

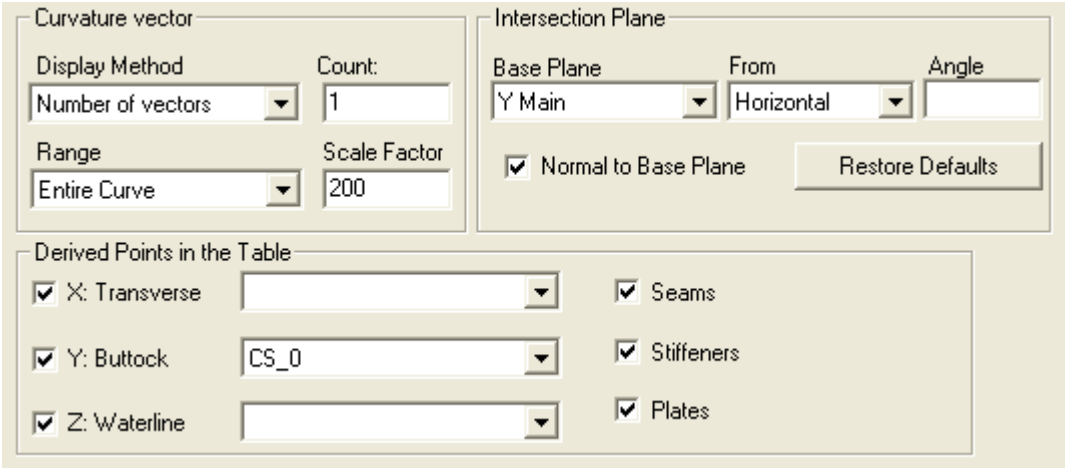
Y Offset - Type an offset value from the Y reference.

Z Reference - Select a waterline (X-Y) grid plane to specify the Z-coordinate of the landing curve point. You can also explicitly specify the Z coordinate by typing the point's location along the Z-axis.

Z Offset - Type an offset value from the Z reference.

TIPS

- Points for which you have entered coordinates are usually definition points.
- Knuckle points cannot be at the ends of a landing curve.
- Every curve must have a minimum of two definition points.



Curvature Vector

- **Display Method** - Specifies how you want the curvature vectors to appear when you click **Preview**. Select **Number of vectors** to specify a fixed number of vectors that the software is to distribute evenly along the landing curve. Specify the number of vectors in the **Count** box. Select **Vector spacing** to place vectors along the landing curve a fixed distance apart. The software determines the number of vectors based on the landing curve length and the spacing distance specified in the **Spacing** box.
- **Count** - Specifies the number of vectors to distribute evenly along the landing curve preview. This option is only available when **Display Method** is set to **Number of vectors**.
- **Spacing** - Specifies the distance between vectors along the landing curve preview. This option is only available when **Display Method** is set to **Vector Spacing**.
- **Range** - Select where along the landing curve preview you want to see curvature vectors. Select **Entire Curve** to display curvature vectors along the entire landing curve. Select **Between Definition Points** to display curvature vectors only between definition points.
- **Scale Factor** - Specify a scaling factor value for the curvature vectors. The larger the value the larger the curvature vectors appear in the graphic view.

Intersection Plane

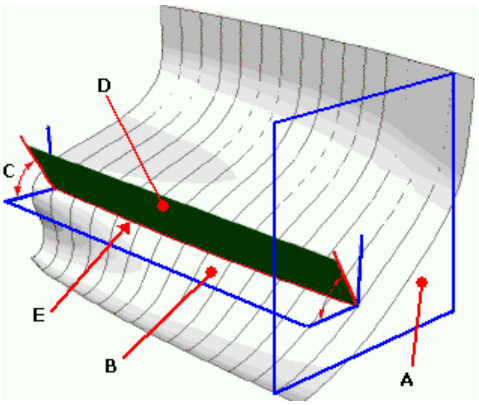
This option is available only when the coordinates of two points are known for a landing curve on a plate system with curvature (such as the hull). When the coordinates of only two points are known, multiple solutions exist to define the landing curve. You can define an intersection plane passing through the points to complete the landing curve definition.

- **Base Plane** - Specifies the plane *in* which to measure the angle used to define the intersection plane.
- **From** - Specifies the plane *from* which to measure the angle used to define the intersection plane. Select **Horizontal** or **Vertical**.
- **Angle** - Specifies the angle of the intersection plane, measured from the **Horizontal** or **Vertical** plane. Use the unit of measure selected in **Tools** > **Options**.
- **Normal to Base Plane** - Select to orient the intersection plane normal to the base plane. The intersection plane appears as a straight line when viewed in the direction of the **Base Plane**. Clear to orient the intersection plane by the minimum linear distance between the two points as allowed by the specified **Angle**.

TIPS

- **Base Plane** is similar to the angle **Plane** in **Profile Orientation Definition**.
- **From** plane is similar to the **Reference** plane in **Profile Orientation Definition**.
- **Angle** is similar to the **Angle** in **Profile Orientation Definition**.
- You can create a profile with a planar web if the **Angle** for the landing curve is the same as the **Angle** for the profile orientation.

In the following example, **A** is the **Base Plane**, **B** is the **From Horizontal** plane, **C** is the **Angle**, **D** is the final **Intersection Plane**, and **E** is the landing curve on the selected plate system.



Derived Points in the Table - These options control when and how the software derives points between definition points that you define. Derived points are not saved as part of the landing curve definition.

- **X: Transverse** - Select this option to have the software derive points where the landing curve intersects each X grid plane. You must select a coordinate system.
- **Y: Buttock** - Select this option to have the software derive points where the landing curve intersects each Y grid plane. You must select a coordinate system.
- **Z: Waterline** - Select this option to have the software derive points where the landing curve intersects each Z grid plane. You must select a coordinate system.
- **Seams** - Select this option to have the software derive points at the intersection of the landing curve with seams on the selected plate system.
- **Stiffeners** - Select this option to have the software derive points at the intersection of the landing curve with stiffeners on the selected plate system.
- **Plates** - Select this option to have the software derive points at the intersection of the landing curve with other plate systems bounded or split by the selected plate system.

Preview - Displays the landing curve, as currently defined by the points in the table, in the model. You can select **Preview** multiple times to see changes made to the landing curve as you add and edit points in the table.

Print - Prints a copy of the landing curve table, as it is currently defined.

Close - Closes the dialog box.

Right-Click Menu Commands

There are several right-click commands available to help in defining the points in the table.

Cut Cell - Removes the contents of the selected cell and moves the contents to the Clipboard.

Copy Cell - Moves a copy of the selected cell contents to the Clipboard.

Paste Cell - Places a copy of the Clipboard contents in the cell.

Cut Row - Removes the contents of the selected row and moves the contents to the Clipboard.

Copy Row - Moves a copy of the selected row contents to the Clipboard.

Paste Row - Places a copy of the Clipboard contents in the row, replacing any existing values in the row.

Paste Reverse - If more than one row is copied, this command places a copy of the Clipboard contents in the same number of rows but in reverse order, replacing any existing values in the rows.

Insert Cut/Copied Rows - Inserts the row on the Clipboard into a new row in the table.

Insert Reverse Rows - If more than one row is copied, this command inserts the rows from the Clipboard in the table in reverse order.

Delete - Removes the contents of the selected cell.

Insert Row - Inserts a new, blank row into the table above the selected row.

Remove Row - Removes the selected row from the table.

Make Derived - Marks the selected reference as the one you want the software to derive.

What do you want to do?

- *Select intersecting object* (on page 764)
- *Set coordinate by plane* (on page 764)
- *Set coordinate by key-ins* (on page 765)
- *Derive points* (on page 765)

Select intersecting object

1. Select the **Point Type**.
2. Click in the **Intersecting Object** cell.
3. Click the browse button  .
4. Select an intersecting object, such as a seam, stiffener, or plate system, to use.

   TIP Only valid intersecting objects are available, based on the relationship to the selected plate system.
5. Click **OK**.

Set coordinate by plane

1. Select the **Point Type**. The valid options are **Definition** and **Knuckle**.
2. Click in the **X**, **Y**, or **Z Reference** cell for the coordinate.
3. Click the browse button  .
4. Select **Reference Plane**.
5. Select a grid plane to use as the coordinate reference.

   TIP Only valid grid planes are available, based on the coordinate reference axis.
6. Click **OK**.

7. Type an offset from the plane for the coordinate, if needed, in the **X**, **Y**, or **Z Offset** box.

Set coordinate by key-ins

1. Select the **Point Type**. The valid options are **Definition** and **Knuckle**.
2. Click in the **X**, **Y**, or **Z Reference** cell for the coordinate.
3. Click the browse button  .
4. Select **Coordinate**.
5. Type the coordinate value in the box.

   TIP Include the unit of measure, such as **m** for meter, if the coordinate value is not in the default unit of measure as defined on the **Units of Measure** tab of **Tools** > **Options**.
6. Click **OK**.

Derive points

1. In row 1, define one end point of the landing curve. Be sure to select the **Definition** option in the **Point Type** cell.
2. In row 2, define the other end point of the landing curve. Be sure to select the **Definition** option in the **Point Type** cell.
3. Add other **Definition** or **Knuckle** points between the end points, if needed.

   TIP Add rows by using the **Insert Row** command on the using right-click menu.
4. In **Derived Points in the Table**, select the axis, axes, or object types to derive points for. In general, you want select an axis or object type that intersects your landing curve.
5. Select the coordinate system or object in the model along which to derive the points.
6. Click **Preview**.
7. Review the points in the table that the software derived. The derived points have **Display** as the **Point Type**.

   NOTES

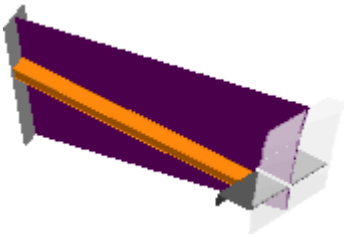
   - You can change a derived point to be a definition point by selecting the **Definition** option in the **Point Type** cell.
   - Click **Preview** when you make any changes to the definition points in the table.

Tripping Stiffener System

The **Tripping Stiffener System** command places a stiffener profile system on a parent plate system, but the stiffener has the following unique characteristics:

- Defined by and bounded to a starting object and an ending object. The objects can be profile stiffener systems, edge reinforcements, plate systems, or edges of the parent plate system.
- Oriented in relation to the orientation of the starting or ending object.

- At least one of the starting and ending objects must penetrate or be bounded to the parent plate of the tripping stiffener.



The software uses the parent plate's molded conventions for the tripping stiffener. Like other profile systems, the tripping stiffener can be split, detailed, and manufactured.

Tripping Stiffener System Ribbon

Displays the options that you use to place a tripping stiffener in the model.

Tripping Stiffener Properties

Opens the **Tripping Stiffener Properties** dialog box, which you use to view and modify the properties of the stiffener system that you are about to place in the model. The initial settings default from the parent plate system. For more information, see *Tripping Stiffener Properties Dialog Box* (on page 770).

Select Plate System

Specifies the plate system to stiffen.

Select Starting Object

Specifies a reference object to serve as the starting location of the tripping stiffener. If the reference object intersects the parent plate system, the software uses the intersection of the two objects as the connect point. If the reference object is an edge of the parent plate system or a profile system that is a child of the parent plate system, then the software uses the mouse click location along the edge or profile as the connect point.

Select Ending Object

Specifies a reference object to serve as the ending location of the tripping stiffener. If the reference object intersects the parent plate system, the software uses the intersection of the two objects as the connect point. If the reference object is an edge of the parent plate system or a profile system that is a child of the parent plate system, then the software uses the mouse click location along the edge or profile as the connect point.

Finish

Places the tripping stiffeners using the defined parameters.

Finish Mode

When set to **On**, places the tripping stiffeners in the model as soon as you set all of the required inputs. When set to **Off**, you must click **Finish** to place the system.

Orientation Table

Opens a read-only table containing the coordinates, orientation angles, and twist rates of the current tripping stiffener system part.

Connect Type

Specifies how the tripping stiffener relates to the selected reference object. You can select from:

- **Connected** - The tripping stiffener connects to and is trimmed by the surface of the reference object. This is the default selection.
- **Lapped** - The tripping stiffener overlaps the reference object.
- **Parallel** - The tripping stiffener landing curve direction is constrained to be parallel to the web of the starting reference object. Not available if the starting and ending reference objects both intersect the parent plate system.
- **Perpendicular** - The tripping stiffener landing curve direction is constrained to be perpendicular to the ending reference object. Not available if the starting and ending reference objects both intersect the parent plate system.
- **Transverse**, **Vertical**, or **Longitudinal** - The tripping stiffener landing curve is constrained to be parallel to one of the main axes, relative to the starting object connect point location. Not available if the starting and ending reference objects both intersect the parent plate system.
- **Fixed X**, **Fixed Y**, or **Fixed Z** - The tripping stiffener end location is defined by a distance from a main axis. Not available if the starting and ending reference objects both intersect the parent plate system.
- **End Connect** - The tripping stiffener connects to the end of the ending reference object when the reference object is a profile stiffener or an edge reinforcement.
- **Distance Along -** The tripping stiffener end location is defined by a distance from a reference object along the ending object.
- **Twisted** - Twist is distributed evenly along the girth length of the tripping stiffener region based on the orientation of the start and end objects. The twist rate is constant. This connect type is available only when both the supports are non-aligned.

NOTE If the tangents computed at the intersecting points of the plate to stiffen and supports are not co-planar, then both of the supports are said to be non-aligned.

Fixed Value

Specifies the value for the Fixed X, Fixed Y, or Fixed Z connect type.

Select Reference Object

Specifies a plate system, profile system, grid plane, seam or reference curve which has a valid intersection with the ending object of the stiffener. The constraint distance is measured along the ending object from the intersection of this reference object and ending object. This option is available only when the **Distance Along** connect type is used for the ending object.

Method

Specifies the constraint direction. In the current release of the software, **Girth** is the only supported constraint method.

Offset

Specifies the constraint distance value for the **Distance Along** connect type. You can type positive and negative values to measure in different directions.

Override Offsets

Ignores the rule-based U and V offsets for the start and ending objects of the tripping stiffener, and sets the offsets to zero and the override flag to **True**.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Section Type

Specifies the profile cross-section type.

Section Size

Specifies the profile section size.

Primary Orientation

Defines the side of the plate system used to place the stiffener profile. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Secondary Orientation

Defines the direction of the web thickness for symmetrical cross-sections and the direction of the flange and web thickness for unsymmetrical cross- sections. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Start Connection

Defines the method of connecting the beginning of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

End Connection

Defines the method of connecting the end of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

What do you want to do?

- *Place tripping stiffener* (on page 769)
- *Modify a tripping stiffener profile* (on page 769)
- *Delete a tripping stiffener profile* (on page 770)

Place tripping stiffener

1. Click **Tripping Stiffener System** on the vertical toolbar.
2. Select the parent plate system to stiffen.
3. Select a **Connect Type** for the tripping stiffener starting object.
4. Select a plate system, profile system, or parent plate edge to use as the starting object for the tripping stiffener.
5. Select a **Connect Type** for the tripping stiffener ending object.
6. Select a plate system, profile system, or parent plate edge to use as the ending object for the tripping stiffener.
7. Click **Finish**.

IMPORTANT At least one of the starting and ending objects must penetrate or be bounded to the parent plate system of the tripping stiffener.

NOTE Tripping stiffener profiles are always associated with the parent plate system to which they belong.

Modify a tripping stiffener profile

1. Click **Select** on the vertical toolbar.
2. Select **Root Profiles** in the **Locate Filter**.
3. Select the tripping stiffener profile to modify.

4. Using the ribbon controls, modify the tripping stiffener profile as needed.

Delete a tripping stiffener profile

1. Click **Select** on the vertical toolbar.
2. Select **Root Profiles** in the **Locate Filter**.
3. Select the tripping stiffener profile to delete.
4. Click **Delete**.

Tripping Stiffener Properties Dialog Box

Specifies the properties for the root or leaf tripping stiffener profile system that you are editing.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Profile Properties Dialog Box) (on page 730)

Cross Section Tab (Profile Properties Dialog Box) (on page 732)

Section Orientation Tab (Profile Properties Dialog Box) (on page 732)

Tripping Stiffener Connection Data Tab (Tripping Stiffener Properties Dialog Box) (on page 771)

Configuration Tab (on page 124)

General Tab (on page 125)

Tripping Stiffener Connection Data Tab (Tripping Stiffener Properties Dialog Box)

Specifies the tripping stiffener connection data as it relates to the two reference objects that define the stiffener location. This tab is available only when you are modifying a tripping stiffener.

Connect Type - Specifies how the tripping stiffener relates to the selected reference object. You can select from:

- **Connected** - The tripping stiffener connects with a butt or tee type connection and is trimmed by the reference object.
- **Lapped** - The tripping stiffener overlaps the reference object.
- **Fixed X**, **Fixed Y**, or **Fixed Z** - The tripping stiffener end location is defined by a distance from a main axis.
- **Parallel** - The tripping stiffener landing curve direction is constrained to be parallel to the web of the beginning reference object.
- **Perpendicular** - The tripping stiffener landing curve direction is constrained to be perpendicular to the ending reference object.
- **Transverse**, **Vertical**, or **Longitudinal** - The tripping stiffener landing curve is constrained to be parallel to one of the main axis.

Load Point - Specifies the load point of the beginning and ending reference object to which the tripping stiffener landing curve is to connect. You must select the **Override** option to change this value.

U Offset - Specifies an offset of the tripping stiffener landing curve relative to the load point on the cross-section of the reference profile. The offset value is based on the approach angle between the end objects.

V Offset - Specifies an offset of the tripping stiffener landing curve relative to the load point on the cross-section of the reference profile. The offset value is based on the approach angle between the end objects.

Override - Select to override the values of **Load Point**, **U Offset**, and **V Offset**.

Profile Edge Reinforcement

The **Profile Edge Reinforcement** command places a profile system along the free edge of a plate system. A free edge is defined as a plate edge that has no logical connections. Therefore, a free edge can be an opening edge, an edge made by use of a boundary curve, or an edge defined by use of a grid plane.

NOTE When an edge reinforcement profile system is created, an edge reinforcement part is also created. Profile parts are detailed in the **Structural Detailing** task and then used to create manufacturing parts in the **Structural Manufacturing** task. Properties of the part can be modified. For more information, see *Edge Reinforcement Profile Part Properties Dialog Box* (on page 779).

Profile Edge Reinforcement Ribbon

Displays the options that you use to place a profile edge reinforcement in the model.

Edge Reinforcement System Properties

Opens the **Edge Reinforcement Properties** dialog box, which you use to view and modify the properties of the edge reinforcement system that you are about to place in the model. For more information, see *Edge Reinforcement Properties Dialog Box* (on page 775).

Pick Edge

Specifies the plate edge to reinforce.

Define Orientation

Specifies the orientation of the edge reinforcement. See the *Define Orientation Controls* section below.

Boundary 1

Defines the first boundary for the edge reinforcement.

Boundary 2

Defines the second boundary for the edge reinforcement.

Edge Boundary

Allows you to select the edge of a plate as the boundary. This option is available when you select **Boundary 1** or **Boundary 2**.

Finish

Places the edge reinforcement using the defined parameters.

Define Orientation Controls

Available when you use the **Define Orientation Definition** option.

Rule

Specifies the orientation rule. This option is only available after you click **Define Orientation**. You can select from the following rules:

Normal - The web of the edge reinforcement system part follows the plate system surface normal along the landing curve. This is the default setting.

Vertical - The orientation vector of the edge reinforcement cross-section remains in a vertical direction. For example, you can select **Vertical** for edge reinforcements on a deck.

Longitudinal - The orientation vector of the edge reinforcement cross-section remains in a longitudinal direction. For example, you can select **Longitudinal** for edge reinforcements on a transverse frame.

Transverse - The orientation vector of the edge reinforcement cross- section remains in a transverse direction. For example, you can select **Transverse** for longitudinal stringer edge reinforcements.

Planar - The orientation angle remains constant relative to the **Normal** reference plane. This rule is valid only for planar edges.

Angle Reference

Specifies the angle reference plane from which to measure the mounting angle. This option is available only when you select **Planar** from the **Rule** list. You can select from the following reference planes.

Normal - Specifies that the reference plane from which the mounting angle is measured is in the Normal direction.

Up-Down - Specifies that the reference plane from which the mounting angle is measured is in the Up-Down direction.

Fore-Aft - Specifies that the reference plane from which the mounting angle is measured is in the Fore-Aft direction.

Port-Starboard - Specifies that the reference plane from which the mounting angle is measured is in the Port-Starboard direction.

Angle

Specifies the angle for the plane. This is set as the mounting angle for the edge reinforcement. The allowable range for the angle is -90 to +90 degrees. The **Angle** option is only available when you select **Planar** in the **Rule** list.

Reject

Clears all selected edges or boundaries.

Accept

Accepts all selected edges or boundaries.

Section Type

Specifies the profile cross-section type.

Section Size

Specifies the profile cross-section size.

Name

Specifies the name of the object that you are placing. Type a name, or leave the box blank to use the default naming rule.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Section Type

Specifies the profile cross-section type.

Section Size

Specifies the profile section size.

Primary Orientation

Defines the side of the plate system used to place the stiffener profile. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Secondary Orientation

Defines the direction of the web thickness for symmetrical cross-sections and the direction of the flange and web thickness for unsymmetrical cross- sections. The default setting is determined by the molded conventions of the parent system. This property can be modified only at the root system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Start Connection

Defines the method of connecting the beginning of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

End Connection

Defines the method of connecting the end of the profile to its plate, profile, or member boundary. **By Rule** is the default setting. The connection is determined by assembly connection rules defined for Structural Detailing. Other connection methods can be added as part of your reference data customization.

By Rule - The connection is determined by assembly connection rules defined for Structural Detailing

Welded - Places a welded web and flange cut.

Snip - Places a snipped web and flanged cut.

Cutback - Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed - Places a bracket as a child of the assembly connection.

What do you want to do?

- *Place edge reinforcement profile* (on page 774)
- *Modify an edge reinforcement profile* (on page 775)
- *Delete edge reinforcement profile* (on page 775)

Place edge reinforcement profile

1. Click **Profile Edge Reinforcement** on the vertical toolbar.
2. Select the edge to reinforce.
3. Click **Define Orientation**.
4. Define the orientation of the profile.
5. Click **Boundary 1**, and define the first boundary for the profile.

NOTE To select the edge of a plate as the first boundary, click **Edge Boundary** .

6. Click **Boundary 2** , and define the first boundary for the profile.

NOTE To select the edge of a plate as the second boundary, click **Edge Boundary** .

7. Click **Accept** .
8. Click **Finish**.

NOTE Profiles are always associated with the plate systems to which they belong.

Modify an edge reinforcement profile

1. Click **Select** on the vertical toolbar.
2. Select **Root Profiles** in the **Locate Filter**.
3. Select the edge reinforcement profile to modify.
4. Click **Edit** > **Properties**.
5. Modify the profile as needed.

Delete edge reinforcement profile

1. Click **Select** on the vertical toolbar.
2. Select **Root Profiles** in the **Locate Filter**.
3. Select the edge reinforcement profile to delete.
4. Click **Edit** > **Delete**.

Edge Reinforcement Properties Dialog Box

Specifies the settings for the profile edge reinforcement.

Main Tab (Edge Reinforcement Properties Dialog Box) (on page 775)

Cross Section Tab (Profile Properties Dialog Box) (on page 732)

Section Orientation Tab (Edge Reinforcement Properties Dialog Box) (on page 777)

Configuration Tab (on page 124)

Relationship Tab (on page 123)

General Tab (Edge Reinforcement Properties Dialog Box) (on page 779)

Main Tab (Edge Reinforcement Properties Dialog Box)

Specifies the general settings for the profile edge reinforcement. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare

name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Specifies how the name is created.

- Select **StdProfileSystemNamingRule** to use the following syntax: <Parent Name>-<Index Number><"X">-<Index Number>. For example, L3-1LBH-1-1X-1, where L3-1LBH-1 is the parent name, 1X is an index number appended by the letter X, and 1 is an index number.
- Select **User Defined** if you want to name the profile yourself by using the appropriate box.

Type

Displays the object type, which in this case is Edge Reinforcement.

Naming Category

Select a naming category. The category is used by the naming rule to name the profile part that is a child to the profile system.

Parent System

Displays the name of the parent system of the edge reinforcement profile.

Description

Type a description for the edge reinforcement profile.

Specification

Select a specification for the edge reinforcement profile.

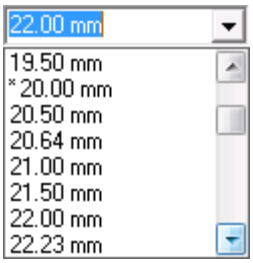
NOTE **Specification** can only be changed on a root edge reinforcement system. This option is unavailable for the leaf system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

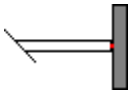
Section Orientation Tab (Edge Reinforcement Properties Dialog Box)

Specifies the position of the cross-section relative to the free edge.

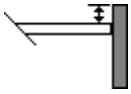
NOTE The Primary Orientation is the only option that you can change on root edge reinforcement systems. You can change all the other options on leaf edge reinforcement systems.

Position - Specifies how the cross section of the profile is attached to the free edge.

On Edge - Centered indicates that the cross-section is centered on the plate thickness using the cross-section load point that you select.



On Edge - Offset indicates that the cross- section load point is offset from the free edge the distance you specify in the **Offset** box.



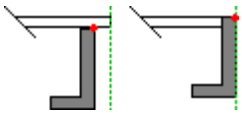
On Face indicates that the cross-section load point is placed on the face of the plate. The load point can be offset from the free edge the distance you specify in the **Offset** box.



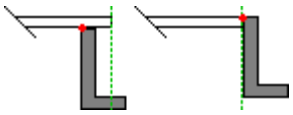
Primary Orientation - Specifies on which side of the plate system the profile is placed. The default setting is the molded conventions of the parent system. The options vary depending on the placement of the edge reinforcement. For example, if the edge reinforcement is running transverse, then the options are **Fore** and **Aft**. This option is not available when **On Edge - Centered** is selected for **Position**.

Secondary Orientation - Specifies the orientation of the web thickness for symmetrical cross-sections and both the web thickness and the flange direction for unsymmetrical cross-sections. The options are:

- **Toward Plate** - The plate edge is moved back so that the flange is at the plate edge.



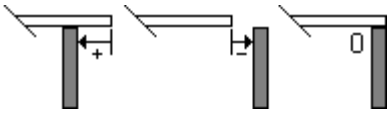
- **From Plate** - The flange is added to the plate edge.



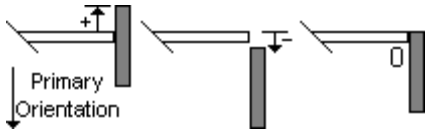
Parent System Type - Displays the object type of the parent system. You cannot change this value.

Offset - Specify the distance that the cross-section is offset from free edge or from the plate face.

- If **Position** is set to **On Face**, then the offset is measured from the free edge. A positive offset value moves the cross- section towards the plate. A negative offset value moves the cross-section away from the plate.



- If **Position** is set to **On Edge**, then the offset is measured from the plate face. A positive offset value moves the cross- section in the opposite direction from the primary orientation. A negative value moves the cross-section in the same direction as the primary orientation.



Mounting Face - Select the cross-section face to mount to the plate system. The options vary depending on the selected profile cross-section. This option is not available when **On Edge - Centered** is selected for **Position**.

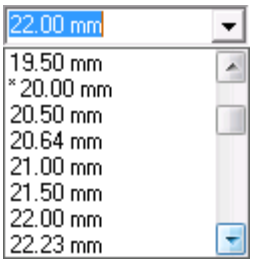
Load Point - Select the cross-section load point through which the free edge passes. This option is not available when **On Edge - Centered** or **On Edge - Offset** is selected for **Position**.

Start Connection Method - Defines the method of connecting the beginning of the profile to its plate, profile, or member boundary. **By Rule** is the default option. The connection is determined by assembly connection rules defined for the Structural Detailing. Other connection methods can be added as part of your reference data customization.

End Connection Method - Defines the method of connecting the end of the profile to its plate, profile, or member boundary. **By Rule** is the default option. The connection is determined by assembly connection rules defined for the Structural Detailing. Other connection methods can be added as part of your reference data customization.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

General Tab (Edge Reinforcement Properties Dialog Box)

Specifies general properties of the edge reinforcement profile system. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

ContinuityType - Select the continuity type for the profile system. Continuity defines how the profile system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the profile system penetrates the other system. Select **Intercostal** to indicate that the profile system is penetrated by the other system.

flangeDirection

Specifies the direction of the web thickness for symmetrical crossing-sections or the direction of the flange and web thickness for unsymmetrical cross-sections.

NamingCategory - Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Edge Reinforcement Properties Dialog Box)* (on page 775).

orientation

Specifies the orientation of the edge reinforcement.

Displays the relative continuity priority. For example, if a continuous part of priority 1 intersects a continuous part of priority 2, the priority 2 part is intercostal at the intersection.

ProfileType - Displays the profile type.

webThicknessOffset

Specifies the offset direction along the plate face.

webThicknessOrientation

This property is not used.

Edge Reinforcement Profile Part Properties Dialog Box

Specifies the properties for the edge reinforcement profile part that you are editing. Default properties are inherited from the parent system of the part.

Main Tab (Edge Reinforcement Profile Part Properties Dialog Box) (on page 780)

Cross Section Tab (on page 781)

Section Orientation Tab (Edge Reinforcement Profile Part Properties Dialog Box) (on page 781)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (Edge Reinforcement Profile Part Properties Dialog Box) (on page 782)

Weight & CG Tab (on page 129)

Extended User Attributes Tab (on page 129)

Main Tab (Edge Reinforcement Profile Part Properties Dialog Box)

Specifies the general properties of the edge reinforcement profile part.

Name

Specifies the name of the edge reinforcement profile part. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the edge reinforcement profile part.

- Select **StiffenerPartRule** to create a name based on the parent root profile system name, the location within the parent root profile system, and the parent assembly name.
- Select **User Defined** to name the part yourself by using the appropriate box.

Type

Displays the type of the part. It is inherited from the parent system.

Naming Category

Select a category for the edge reinforcement profile part. Categories specify the role of the part in the model.

Parent System

Displays the parent system for the edge reinforcement profile part.

Specification

Displays the structural specification for the edge reinforcement profile part. It is inherited from the parent system.

Description

Type a description for the edge reinforcement profile part.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Cross Section Tab

Specifies the cross-section settings for the part.

Section Type

Displays the profile section type.

Section Size

Displays the profile section size.

Material

Displays the material for the part.

Grade

Displays the material grade for the part.

NOTE The tab also displays the graphic and parameters of the cross-section symbol used by the part.

Section Orientation Tab (Edge Reinforcement Profile Part Properties Dialog Box)

Specifies the section orientation settings for the edge reinforcement profile part.

Position

Displays how the cross-section of the profile is attached to the free edge.

Primary Orientation

Displays which side of the plate to place the edge reinforcement profile.

Secondary Orientation

Displays the direction of the web thickness for symmetrical crossing-sections and the direction of the flange and web thickness for unsymmetrical cross-sections.

Parent System Type

Displays the object type of the parent system.

Offset

Displays the distance that the cross-section is offset from free edge or from the plate face.

Mounting Face

Displays the cross-section face to mount to the plate.

Load Point

Displays the cross-section load point through which the landing curve passes.

General Tab (Edge Reinforcement Profile Part Properties Dialog Box)

Specifies general properties of the stiffener part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

flangeDirection

Specifies the direction of the web thickness for symmetrical crossing-sections or the direction of the flange and web thickness for unsymmetrical cross-sections.

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Edge Reinforcement Profile Part Properties Dialog Box)* (on page 780).

orientation

Specifies the orientation of the edge reinforcement.

ProfileType

Specifies the profile type of the part. The list is defined by the StrMfgProfileType codelist.

webThicknessOffset

Specifies the offset direction along the plate face.

webThicknessOrientation

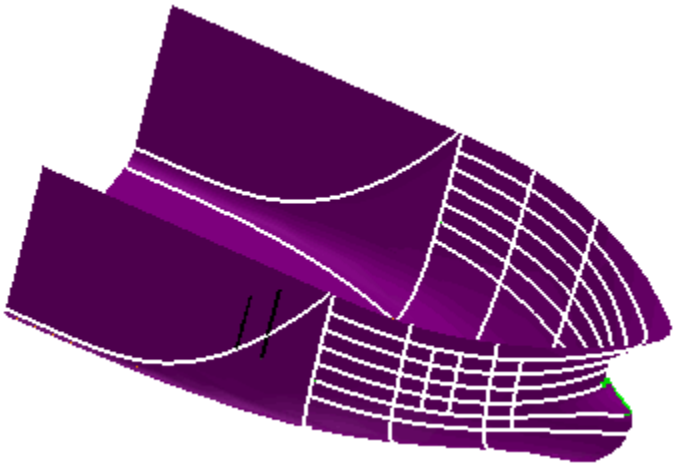
This property is not used.

Modify Profile System Landing Curve Display Colors

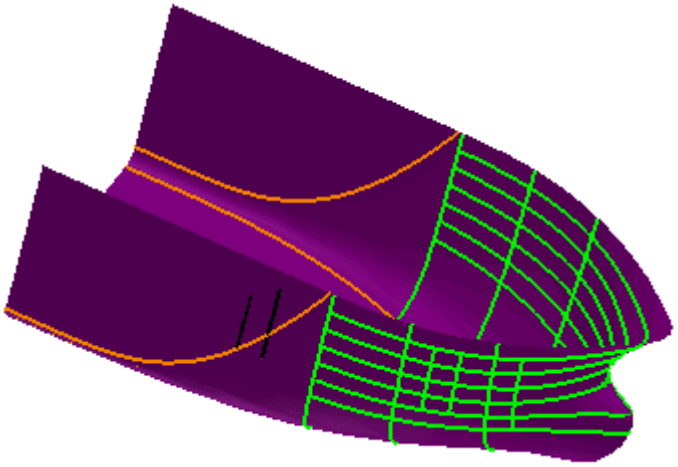
Use this procedure to create style rules to customize the display colors of landing curves. You can use the procedure to set different colors for the landing curves of:

- Seams
- Profile Systems
- Reference Curves
- Plate Knuckle Curves

For example, before creating style rules, reference curves and design seams are white:



After creating separate style rules for reference curves and design seams, they appear in different colors:



Use the following steps to customize curve display colors.

Select the View Aspect

1. Click **Format** >**View**.

   *The **Format View** dialog box opens.*

2. In the **Selected Aspects** box, click **Reference Geometry**.

   NOTE By default, **Simple physical** is also selected.

3. Click **OK**.

Create a Style Rule

1. Click **Format** > **Surface Style Rules**.

   *The **Surface Style Rules** dialog box opens.*

2. Click **New**.

   *The **Surface Style Rule Properties** dialog box opens*

3. Type a name for the new surface style rule in the **Rule name** box.
4. In the **Filter** list, click **More** to create a new filter.

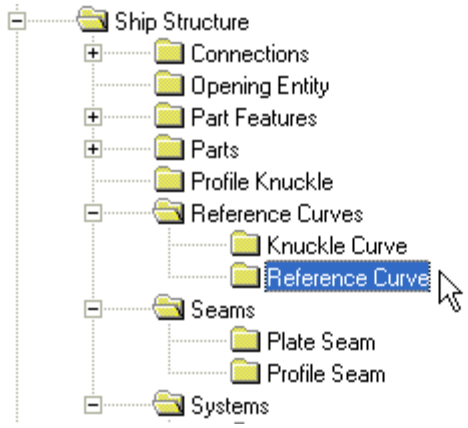
   *The **Select Filter** dialog box opens.*

5. Click **New Filter**.

   *The **New Filter Properties** dialog box opens.*

   NOTE If you are an administrator, you can first select **Ship Filters**. The new filter is then available to all users.

6. On the **Object Type** tab, expand the hierarchy **Ship Structure**, and select one of the following:
   - Ship Structure/ Reference Curves/**Reference Curve**
   - Ship Structure/ Reference Curves/**Knuckle Curve**
   - Ship Structure/Seams/**Plate Seam**
   - Ship Structure/Systems/Profile Systems/**Stiffener System**



7. Click **OK**.
8. Click **OK** in the **Select Filter** dialog box.

9. In the **Surface Style Rule Properties** dialog box, select a color in the **Style applied** box.
10. Click **OK**.

    *The new rule appears in the **Style rule library** list.*

Add the Rule to the Workspace

1. Select the rule in the **Style rule library** list.
2. Click **Add**.
3. Click **OK**.

See Also

In the Common User's Guide:
Format View
Format a View
Surface Style Rules
Create a Surface Style Rule
Create a New Filter
Add a Surface Style Rule to the Workspace

Extend

Extends landing curve definitions in a tangential direction so that an object used to define the landing curve can also be used as a bounding object. Also extends plate systems sketched boundary curves to ensure that a closed boundary is created.

This command is available on the context menu when a sketched profile or seam is selected.

What do you want to do?

- *Extend a profile or seam in one direction* (on page 785)
- *Extend a profile or seam in both directions* (on page 785)
- *Display a profile or seam dynamically while extending* (on page 786)

Extend a profile or seam in one direction

1. Select the sketched profile or seam to extend.

   TIP You can select the profile or seam in the Workspace Explorer or in the graphic view.
2. Right-click, and select **Extend** from the context menu.
3. Move the mouse to indicate the direction to extend the profile or seam.
4. Left-click to accept the extension.

   The software extends the profile or seam to the boundary.

Extend a profile or seam in both directions

1. Select the sketched profile or seam to extend.

   TIP You can select the profile or seam in the Workspace Explorer or in the graphic view.

2. Right-click, and select **Extend** from the context menu.
3. Rotate the mouse wheel to extend the profile or seam.
4. Left-click to accept the extension.

   The software extends the profile or seam to the boundary.

Display a profile or seam dynamically while extending

1. Select the sketched profile or seam to extend.

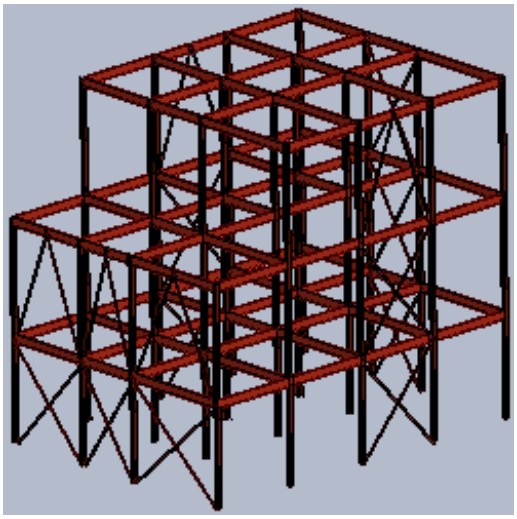
   TIP You can select the profile or seam in the Workspace Explorer or in the graphic view.
2. Right-click, and select **Extend** from the context menu.
3. Press SHIFT, and move the mouse or rotate the mouse wheel to extend the profile or seam. For more information, see *Extend a profile or seam in one direction* (on page 785) or *Extend a profile or seam in both directions* (on page 785).

   The profile or seam displays dynamically.
4. Left-click to accept the extension.

   The software extends the profile or seam to the boundary.

SECTION 7

Place Linear Member Systems



Places a linear member in the model. You can place beams, columns, braces, truss elements, or cable member types using this command. You can place standard members (rolled shapes) or designed members (built-up shapes). Use this command when you want to place members by specifying the exact start and end points.

NOTES

- You can define custom member types by editing the **Structural Member Type** select list in the Catalog task.
- When a linear member system is connected directly to another member, a frame connection is created. If detailed trimming is needed in a plant database, use the **Trim Member** or **Place Assembly Connection**. If detailed trimming of the members is needed in a marine database, use **Place Member Generic Assembly Connection** in the Molded Forms or Structural Detailing task.
- When a linear member system is connected to a plate system or edge reinforcement system, the member is trimmed to the plate and a generic assembly connection is created. In marine mode, the bounding ports can be modified with **Place Member Generic Assembly Connection** in the Molded Forms or Structural Detailing task. For more information, see *Place Member Generic Assembly Connection* (on page 968).

Place Linear Member System Ribbon

Specifies the properties for the member that you are placing. When editing a member part, this ribbon changes. For more information on the properties that are available when you are modifying a member part, see *Modify Linear Member Part Ribbon*.

Member Properties

Activates the *Member System Prismatic Properties Dialog Box* (on page 813). You can use this dialog box to specify additional member properties that you cannot set on the ribbon, such as material, material grade, and end releases. Because changes made on the ribbon, such as to the **Type Category**, can affect or reset properties in the dialog box, we recommend that you set properties in this dialog box after you have made your selections from the ribbon.

Start

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see Members.

End

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see Members.

Enter AMP

Activates the *Advanced Member Positioning Ribbon*, which you can use when you want to place a member at the intersection of three objects and maintain the relationship to those objects. For example, a column that needs to stay over two walls, or bulkheads, on a lower level. You must select the **Align-Default** or **Align-Lapped** frame connections from the **Connection** box to use the advanced member positioning options.

Finish

Click to place the member in the model. **Finish** is active only when **Finish Mode** is selected.

Finish Mode

Specify whether or not **Finish** must be selected to place a member in the model. If the **Finish Mode** is selected, the software places the member in tentative mode after you identify the second end point. This tentative mode allows you to modify placement settings such as the offset, cardinal point, or frame connection properties before you commit the member to the model. If the **Finish Mode** is not selected, then the software automatically places the member in the model after you identify the second end point.

Connection

Select the frame connection type to use for the member that you are placing. If you select **By Rule**, the software determines the frame connection to use based on the geometry between the member that you are placing and existing members in the model. If you select **More**, all available frame connections display from which you can select the frame connection to use. For more information about frame connections, see Frame Connections. This option is not available if you are editing an existing member. You can change the frame connection later by editing the frame connection directly.

Connection Properties

Activates the **Connection Properties** dialog box, which is used to specify properties for the active frame connection. The properties that appear in this dialog box are described under the Connection Properties section.

System

Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* in *Place Linear Member Systems* (on page 787).

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

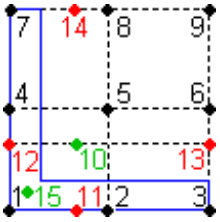
Specifies the type of member, such as a beam or column. The available member properties

change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

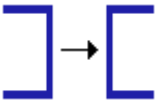


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

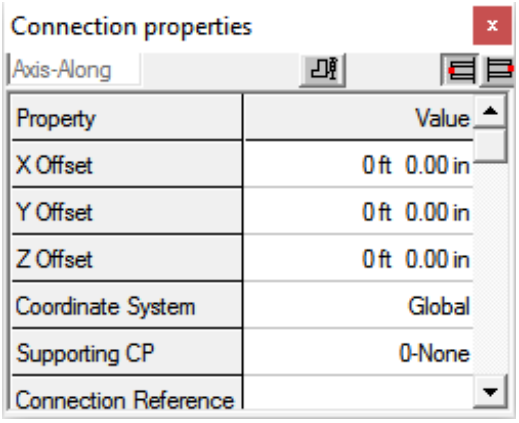
Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the Z-axis or the X-axis of the active coordinate system depending on the member orientation. The active coordinate system is set using the **Coordinate System** control in **PinPoint**  . The **Angle** property is not used when placing members using the **Seated**, **Flush**, or **Centerline** frame connections.

Reflect



Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

Connection Properties



The **Connection Properties** appear only when you have selected the **Connection Properties**  option. Connection properties change depending on the frame connection specified in the

Connection option. To see the frame connection properties for the start of the member, select . To see the frame connection properties for the end of the member, select . Click to see a preview of the frame connection. The frame connection type appears in the upper left corner of the dialog box.

TIP The supported member is the member that you are placing. The supporting member is the existing member in the model to which you are connecting.

Seated, Flush, and Centerline Frame Connection Properties

Side

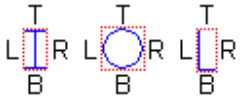
Select the side of the supporting member on which you want to place the supported member.

Offset

Specify the distance to place the supported member from the supporting member. For seated and flush frame connections, the offset is between the side of the supporting member that you specified with the **Side** option and the supported member's side that you specify with the **Edge** option. For centerline frame connections, the offset is between centerline of the supporting member and the supported member's side that you specify with the **Edge** option.

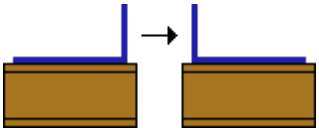
Edge

Specifies the side of the supported member's cross-section that is mated to the supporting member. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.



Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. For example, when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Axis Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction after the two cardinal points are aligned.

Y Offset

Specifies an offset to apply in the y-direction after the two cardinal points are aligned.

Z Offset

Specifies an offset to apply in the z-direction after the two cardinal points are aligned.

Coordinate System

Specifies the coordinate system to use for the offset values. Local is the local coordinate system of the supporting member.

Supporting CP

Specifies to which cardinal point on the supporting member system to align the supported member system's cardinal point. You can specify any cardinal point number, or select 0 to use the cardinal point with which the supporting member was placed.

Surface Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction.

Y Offset

Specifies an offset to apply in the y-direction.

Z Offset

Specifies an offset to apply in the z-direction.

Coordinate System

Specifies the coordinate system to use for the offset values.

Vertical Corner Brace Frame Connection Properties

X Offset

Specifies the offset to apply in the x-direction.

Y Offset

Specifies the offset to apply in the y-direction.

Z Offset

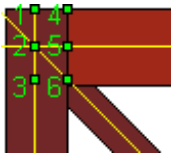
Specifies the offset to apply in the z-direction.

Coordinate System

Select whether the offset values are defined relative to the global coordinate system or the member's local coordinate system.

Work Point

Specifies the work point location. There are six work point locations that you can select.



1 - Primary Center - Secondary Far Side
2 - Primary Center - Secondary Center
3 - Primary Center - Secondary Near Side
4 - Primary Near Side - Secondary Far Side
5 - Primary Near Side - Secondary Center
6 - Primary Near Side - Secondary Near Side

Modify Linear Member Part Ribbon

Displays the member part properties that you are editing when you select one or more linear member parts.

Member Properties

Activates the *Member Part Prismatic Properties Dialog Box* (on page 818). You can use this dialog box to specify additional member part properties, such as material, material grade, and end releases, which you cannot set on the ribbon.

Start

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see

Members.

End

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see Members.

Convert

Translates a member part to have a single stand-alone member system. This option is only available when you select a member part that belongs to a member system that has been split.

When you place a member system, that member system has a single member part associated with it (for more information, see Members). Using the *Place Splits* (on page 921), you can split that single member part into multiple member parts that are each still associated with the original single member system.

If you want to delete or modify only one of the member parts, you must delete the member system and all the other member parts. Deleting a member part deletes its parent member system which causes the sibling member parts to be deleted. However, using the **Convert** option you can cause a member part to have its own, new parent member system. You can then delete or otherwise modify that member part without affecting the other member parts of the original member system.

The software automatically translates the relevant split connections into frame connections. Permission groups of the original member system are used for the new member system.

System

Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* in *Place Linear Member Systems* (on page 787).

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

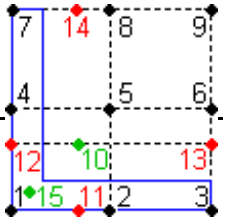
Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

Displays the relative position of the structural cross-section to the



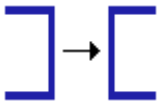
member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the Z-axis or the X-axis of the active coordinate system depending on the member orientation. The active coordinate system is set using the **Coordinate System** control in **PinPoint** . The **Angle** property is not used when placing members using the **Seated**, **Flush**, or **Centerline** frame connections.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.



Advanced Member Positioning Ribbon

Specifies the properties for the member that you are placing using the advanced member positioning option. When editing a member part, this ribbon changes. For more information on the properties that are available when you are modifying a member part, see *Modify Linear Member Part Ribbon* in *Place Linear Member Systems* (on page 787).

Member Properties

Activates the *Member System Prismatic Properties Dialog Box* (on page 813). You can use this dialog box to specify additional member properties that you cannot set on the ribbon, such as material, material grade, and end releases. Because changes made on the ribbon, such as to the **Type Category**, can affect or reset properties in the dialog box, we recommend that you set properties in this dialog box after you have made your selections from the ribbon.

Start

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see Members.

End

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see Members.

Exit AMP

Exits the advanced member positioning ribbon, and returns you to the standard member controls.

Connection Properties

Activates the **Connection Properties** dialog box, which is used to specify properties for the

active frame connection. The properties that appear in this dialog box are described below in the *Connection Properties* section.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

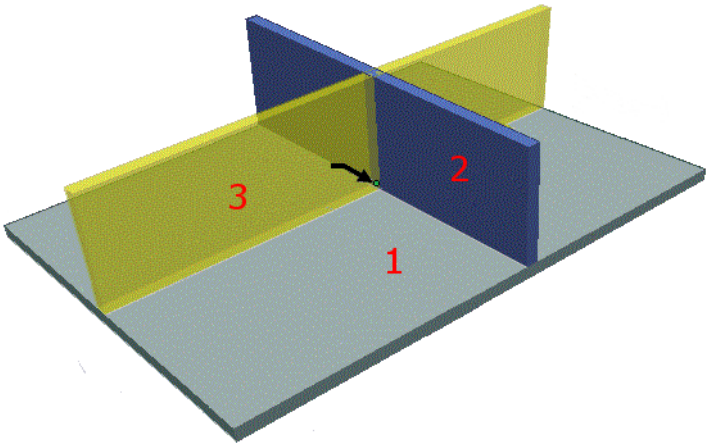
Specifies the bounding object.

2

Specifies the align object.

3

Specifies the intersect object.

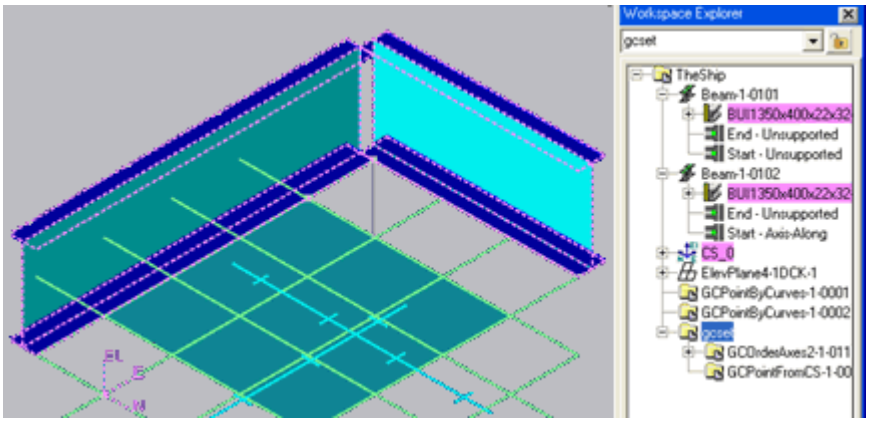


Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

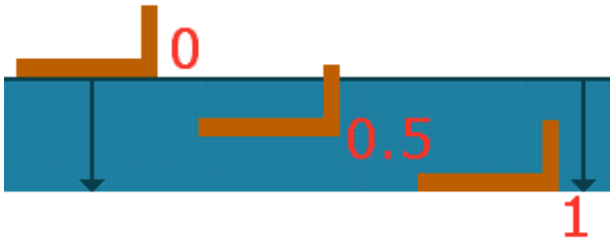
IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Bound Offset

Type an offset distance for the point from the bound surface. This offset is in addition to the ratio you defined.

Bound Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Entering a ratio of 0.5 centers the point between the two surfaces.



Bound Orient

Specifies how to interpret the bounding object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Align Offset

Type an offset distance for the point from the align surface. This offset is in addition to the ratio you defined.

Align Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Entering a ratio of 0.5 centers the point between the two surfaces.

Align Orient

Specifies how to interpret the align object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Intersect Offset

Type an offset distance for the point from the intersect surface. This offset is in addition to the ratio you defined.

Intersect Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Typing a ratio of **0.5** centers the point between the two surfaces.

Intersect Orient

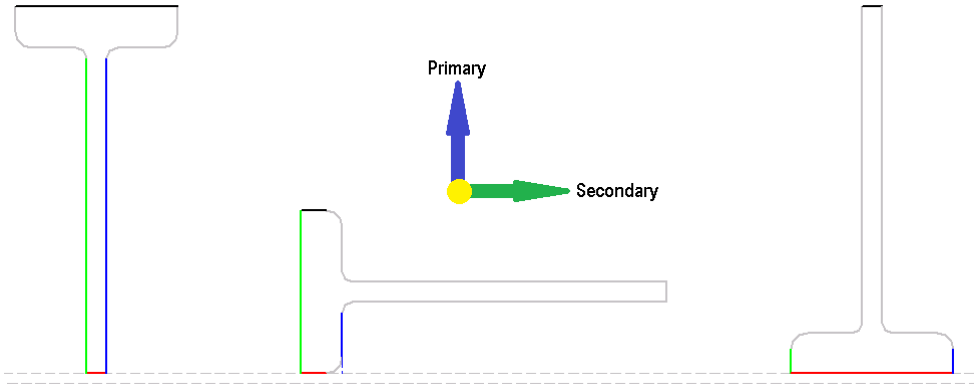
Specifies how to interpret the intersect object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Orient Options

Specifies if the profile is interpreted along its primary orientation or its secondary orientation. If along the primary orientation, then 0.0 is the red surface and 1.0 is the black surface. If along the secondary orientation, then 0.0 is the green surface and 1.0 is the blue surface.

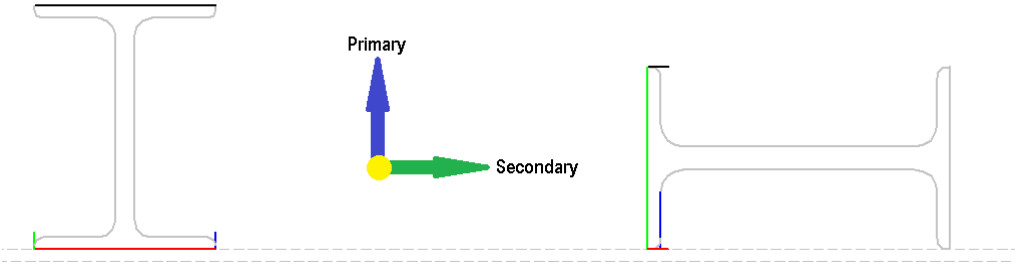
Tee

Includes W, S, and M types and fabricated Ts



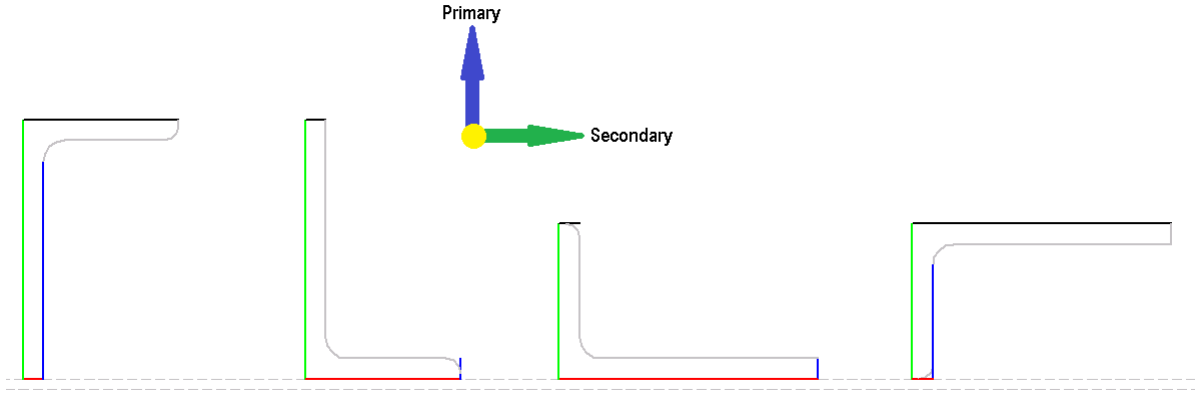
I-beams

Includes W and S types.

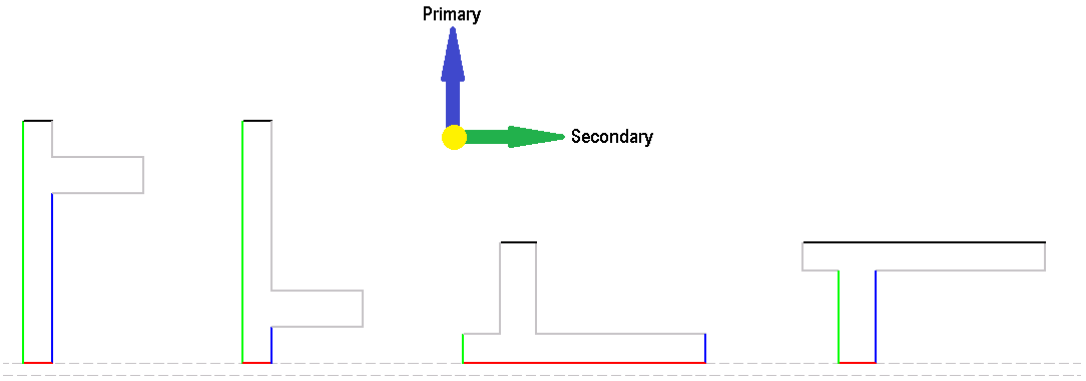


Angle

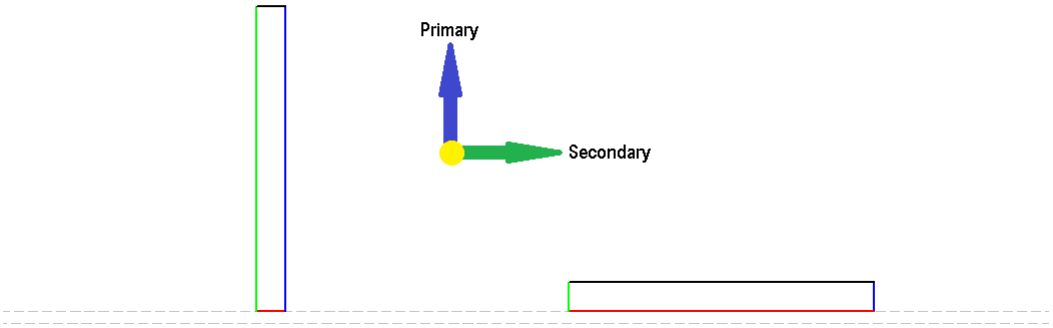
Includes equal and unequal leg angles.



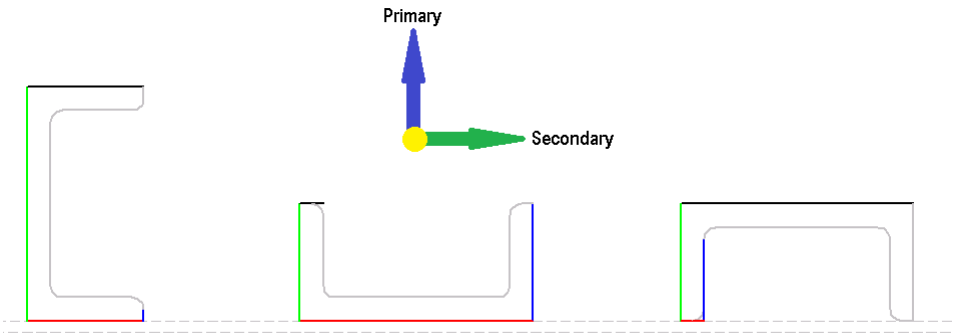
Fabricated Angle



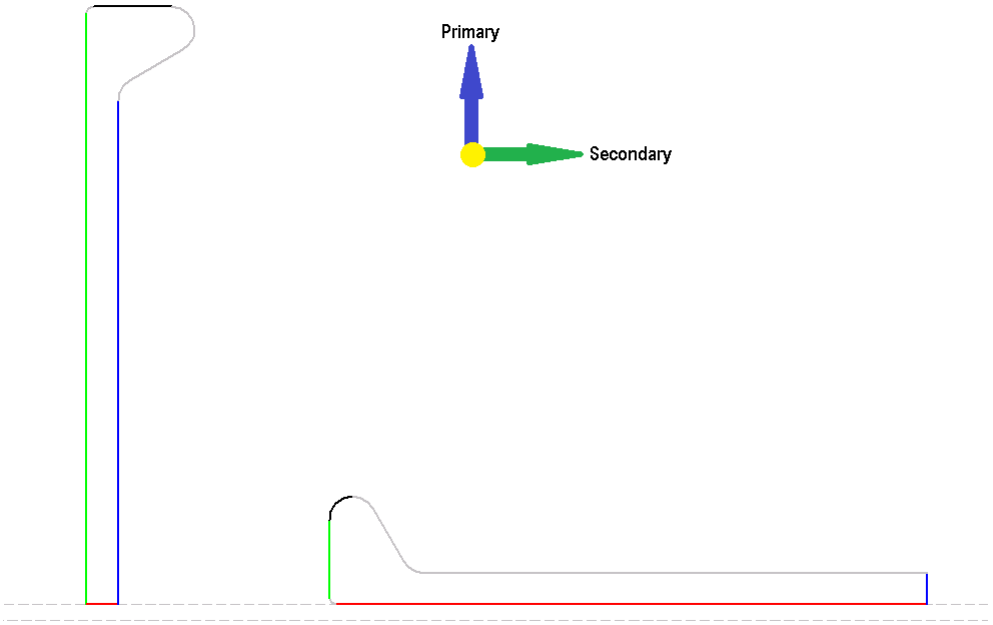
Flat Bar



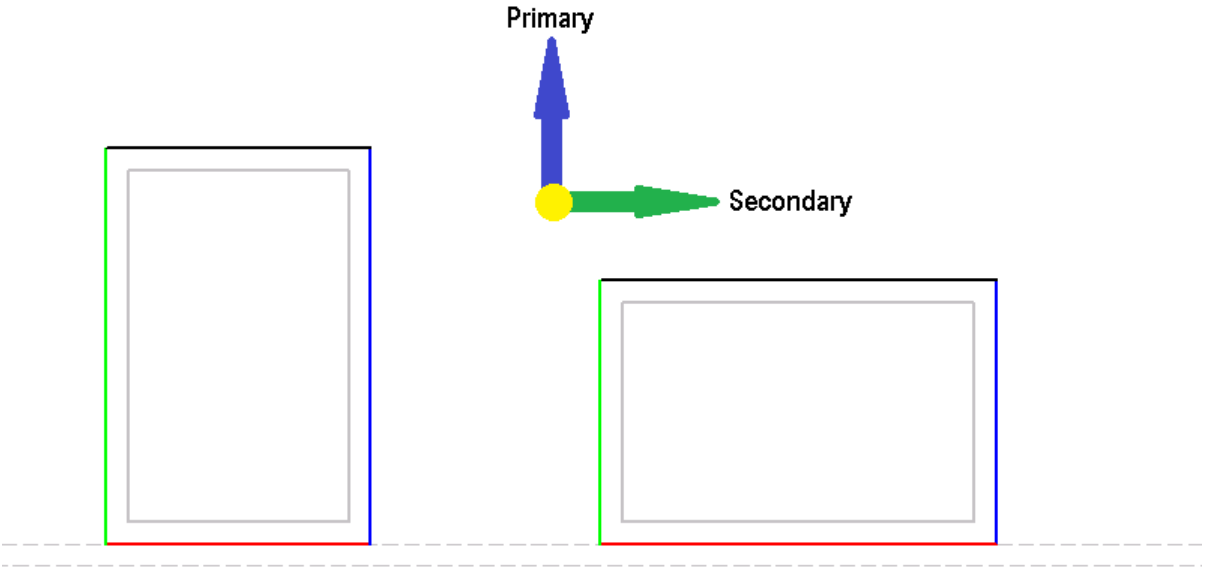
Channel



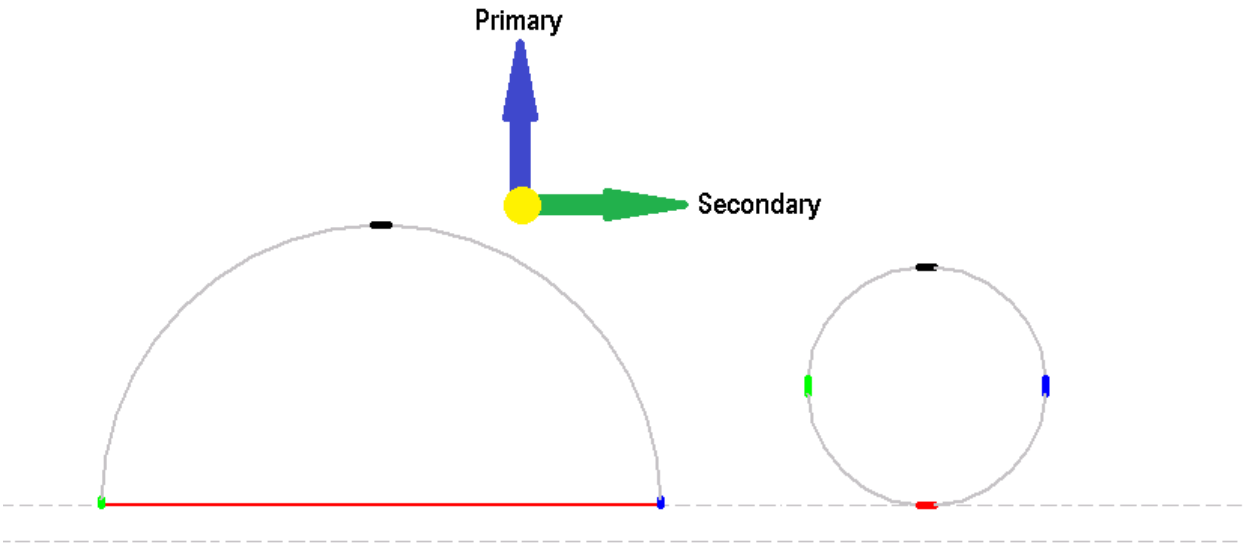
Bulb



Rectangular Tube

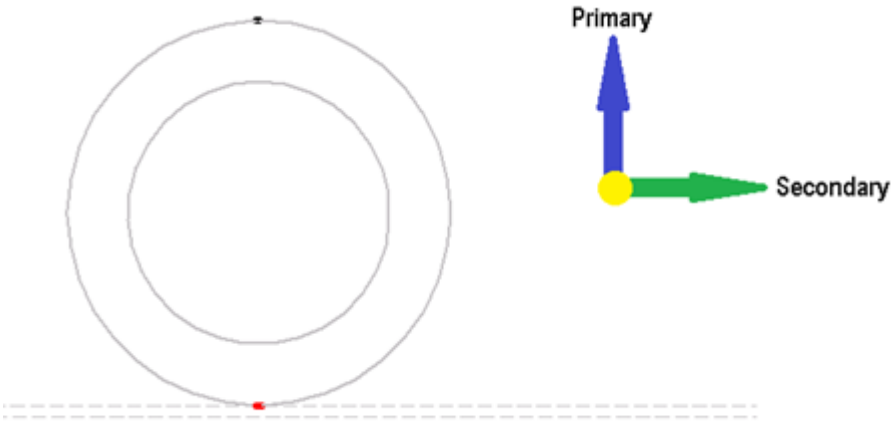


Half Round and Round Bar



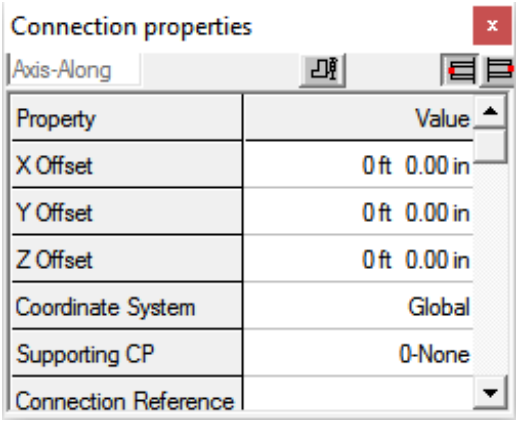
Round Tube (Pipe)

NOTE Round Tube has no faces defined along the secondary orientation.



Connection Properties

The **Connection Properties** appear only when you have selected the **Connection Properties** option. Connection properties change depending on the frame connection specified in the **Connection** option. To see the frame connection properties for the start of the member, select . To see the frame connection properties for the end of the member, select . Click to see a preview of the frame connection. The frame connection type appears in the upper left corner of the dialog box.



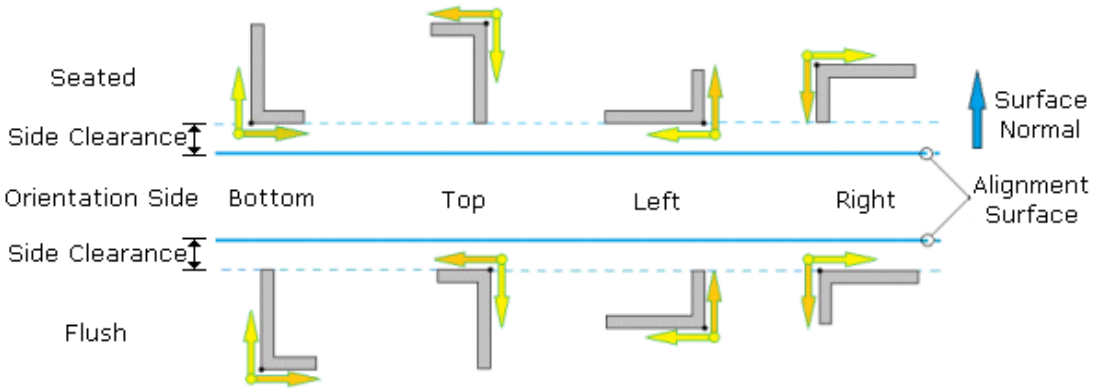
TIP The supported member is the member that you are placing. The supporting member is the existing member in the model to which you are connecting.

Align-Default/Lapped Frame Connection Properties

Side Justification

Select the side of the plane on which to place the member.

- **None** - Aligns the member axis so that it is placed on the surface.
- **Seated** - Sets the member on top of the surface. The selected **Orientation Side** is nearest the surface.
- **Center** - The center of the cross-section is placed on the surface. The center is calculated by the top-bottom, left-right bounds of the cross-section shape.
- **Flush** - Uses the alignment surface's top and bottom extent to position the supported member. The supported member typically lies within the body of the alignment plane with one edge of the member flush with the alignment surface but can be offset.

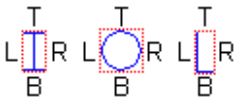


Orientation Control

If **On**, the member angle is controlled by the **Orientation Angle** value defined for the Align frame connection. If **Off**, you can set the angle of the member independently of the align surface by using the **Rotation** property for the member part.

Orientation Side

Specifies the side of the supported member's cross-section that is mated to the alignment surface. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.

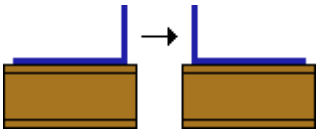


Orientation Angle

When **Orientation Control** is **On**, this angle is added to the selected **Orientation Side** to rotate the member. If the Align frame connection is used on both ends of the member and you define a different angle value for each end, the angle defined at the start end of the member is used.

Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. An example of when to use this option is when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Extend Distance

Specifies the offset distance from the bounding surface along the member's axis.

Side Clearance

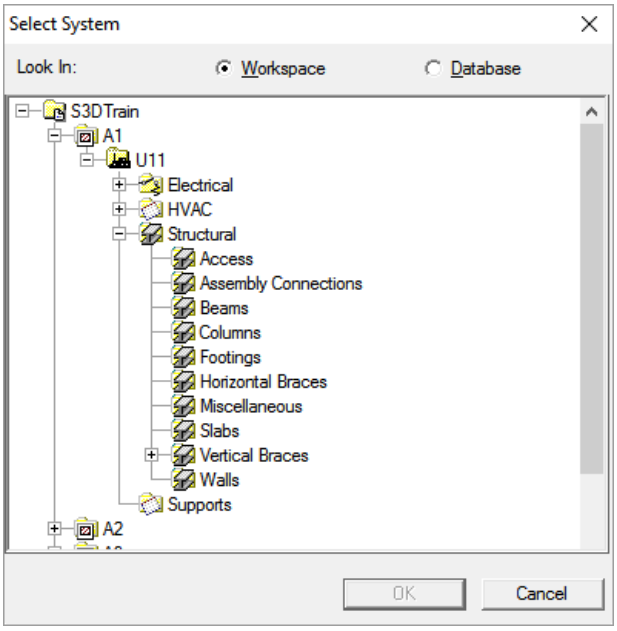
Specify the offset distance perpendicular to the align surface to place the member axis.

Lateral Distance

Specifies the offset distance in the plane of the align surface to place the member axis.

Select System Dialog Box

This dialog box displays when you select the **More...** option in the **System** list. Use this dialog box to select the system that you want. You can create new systems in the Systems and Specifications task.

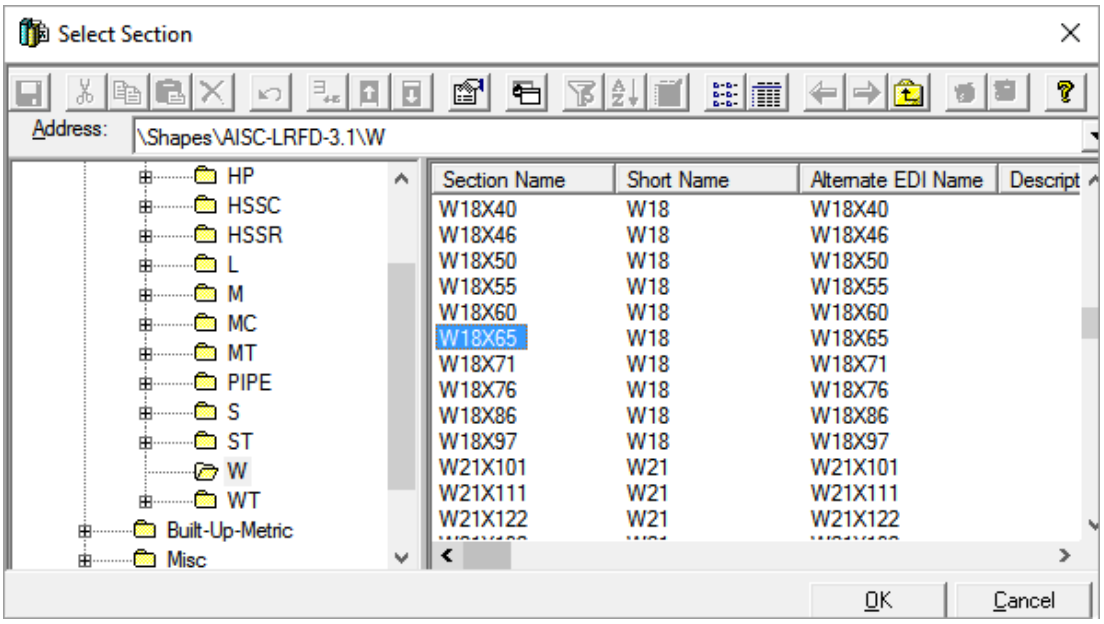


Look in

Specify where you want to look for the system. Select **Workspace** to look for the system in your defined workspace only. Select **Database** to look for the system in the entire Model database.

Select Section Dialog Box

Allows selection of the type of section to be placed. This dialog box appears when you click the **More** option on the **Section Name** list. By browsing through the hierarchy, you can find any section in the Catalog database. After you select a section, the software returns you to the model, where you can finalize placement.



Properties

Displays the properties of the selected section. Because you cannot modify any properties until the section is placed, all properties on the dialog box are read-only.

Preview

Displays a picture of the selected section. The image file must be assigned to the section in the reference data.

Filter

Allows you to filter catalog data to help find the subset of data that you want to work with, similar to Microsoft Excel.

Sort

Sorts the catalog data by column to help you find like items.

Customize Current View

Defines with columns in the data you want to see.

List View

Sets the dialog box to display sections in a list view.

Grid View

Sets the dialog box to display sections in a spreadsheet-style grid view.

Back

Returns you to the previously selected section type or node. Use this command to navigate through the hierarchy to the specific type that you need.

Forward

Sends you to the last selected section type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

Up One Level

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

Address

Specifies your exact location within the displayed hierarchy.

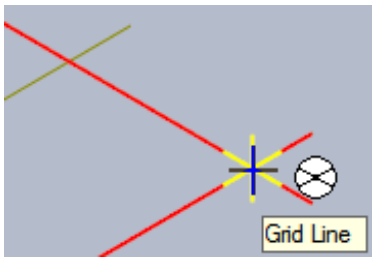
What do you want to do?

- *Place members using discrete placement* (on page 806)
- *Place members using contiguous placement* (on page 807)
- *Place a member using finish mode* (on page 808)
- *Edit member system properties* (on page 809)
- *Edit member part properties* (on page 809)
- *Edit a frame connection* (on page 809)
- *Delete a member system* (on page 809)
- *Convert a member part* (on page 810)
- *Modify the cardinal point of a member* (on page 810)
- *Modify the end releases of a member* (on page 810)
- *Modify the angle of a member* (on page 811)
- *Modify the cross-section of a member* (on page 811)
- *Modify the material of a member* (on page 811)
- *Modify the material grade of a member* (on page 811)
- *Modify the type of member* (on page 812)
- *Move a member* (on page 812)
- *Move one end of a member* (on page 812)
- Move one end of multiple members

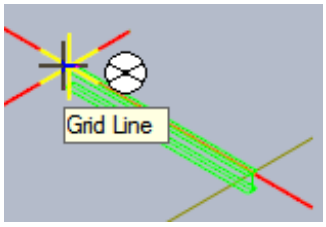
Place members using discrete placement

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

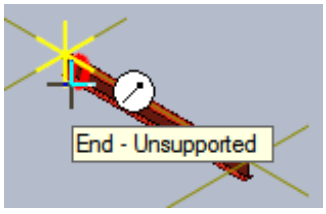
   TIP If you are unsure of which frame connection type to use, review Frame Connections. You can also select the **By Rule** option to allow the software to select automatically a frame connection type.
3. In the **Type category** box, select the member type category to place.
4. In the **Type** box, select the member type to place.
5. Specify the start location, or first point, of the member.

   TIP You can use the frame connection of another member as the start or end location of the member that you are placing.
6. Specify the end location, or second point, of the member.

7. Click **Start** to specify the start location of the next member.
8. Specify the start location of the next member.

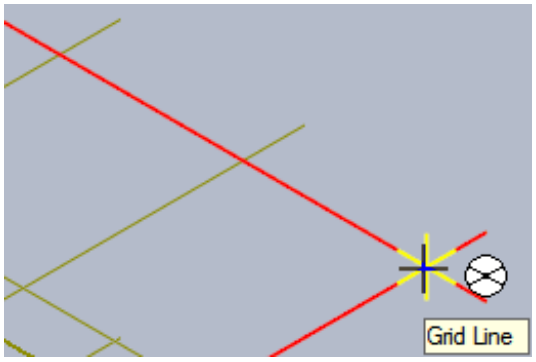
NOTES

- To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** option is selected.
- You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

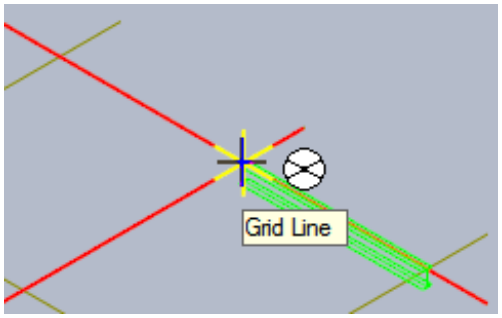
Place members using contiguous placement

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

   TIP If you are unsure of which frame connection type to use, review Frame Connections. You can also select the **By Rule** option to allow the software automatically to select a frame connection type.
3. In the **Type category** box, select the member type category to place.
4. In the **Type** box, select the member type to place.
5. Specify the start location, or first point, of the member.

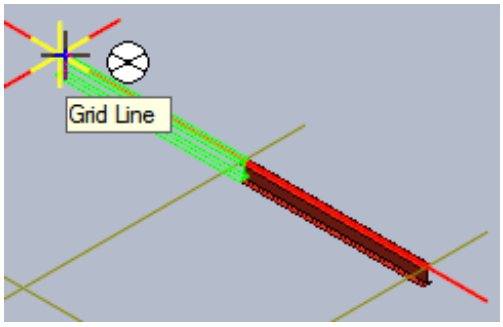
   

   TIP You can use the frame connection of another member as the start or end location of the member that you are placing.
6. Specify the end location, or second point, of the member.

7. Click **End** to activate the contiguous placement mode.

8. Specify the end location of the next member.



NOTE To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** option is selected.

Place a member using finish mode

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

   TIP If you are unsure of which frame connection type to use, review Frame Connections. You can also select the **By Rule** option to allow the software to select automatically a frame connection type.
3. Click **Finish Mode** to activate the **Finish** button.
4. Click **Connection Properties** to activate the **Connection Properties** dialog box.
5. In the **Type category** box, select the member type category to place.
6. In the **Type** box, select the member type to place.
7. Specify the start location, or first point, of the member.

   TIP You can use the frame connection of another member as the start or end location of the member that you are placing.
8. Specify the end location, or second point, of the member.
9. Edit the start and end frame connection properties in the **Connection Properties** dialog box.
10. Edit the member properties using the **Member Properties** dialog box.
11. Click **Finish**.

NOTES

- To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** option is selected.
- You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

Edit member system properties

1. Click **Select** on the vertical toolbar.
2. Select **Members System** in the **Locate Filter**.
3. Select the member system to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed. For more information about properties, see *Member System Prismatic Properties Dialog Box* (on page 813).

Edit member part properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select the member part to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed. For more information about properties, see *Member Part Prismatic Properties Dialog Box* (on page 818).

Edit a frame connection

1. Click **Select** on the vertical toolbar.
2. Select **Frame Connections** in the **Locate Filter**.
3. Select the frame connection to edit.
4. Edit the frame connection as needed. For more information about frame connection properties, see *Frame Connection Properties Dialog Box* (on page 850).

Delete a member system

1. Click **Select** on the vertical toolbar.
2. Select **Members System** in the **Locate Filter**.
3. Select the members to delete.
4. Click **Delete** .

NOTES

- All loads and boundary conditions placed in the Structural Analysis task on the deleted member system are also deleted. This could affect any Analytical Models that have been exported.
- All footings associated with the member system are also deleted.

- Ladders or stairs using the deleted member system as the defined top edge are sent to the **To Do List**.

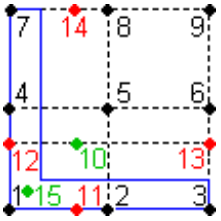
Convert a member part

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member part to have its own member system.
4. Click **Convert** on the ribbon.

Modify the cardinal point of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select the member part to modify.
4. Using the **Cardinal Point** control on the ribbon bar, select a new cardinal point for the member part.

NOTE There are 15 cardinal positions available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13.



Modify the end releases of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Member Properties** on the ribbon.
5. Select the **Member Part** tab.
6. In the **Category** box, select **End Releases**.
7. Modify the member part end releases as required.

Modify the angle of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Angle** control on the ribbon bar, edit the member angle.

   TIP You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

Modify the cross-section of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Section name** box on the ribbon bar, type the new cross-section name for the member, or select **More** to select the cross-section from the Catalog.

Modify the material of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Edit > Properties**.
5. Select the **Cross Section** tab.
6. Select a new value for the **Material** property.

Modify the material grade of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Edit > Properties**.
5. Select the **Cross Section** tab.
6. Select a new value for the **Grade** property.

Modify the type of member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Type Category** and **Type** controls on the ribbon bar, change the type of the member part.

NOTE If the member type category or type that you want to use is not available, you can add it by editing the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook and bulkloading it into the Catalog database. For more information on editing the workbook and bulk loading, see the *Smart 3D Reference Data Guide*.

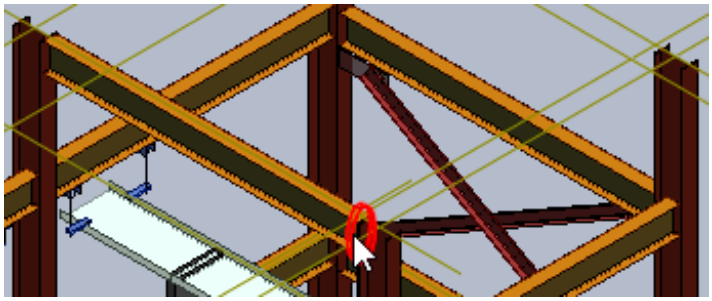
Move a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to move.
4. Click **Move** on the main ribbon.
5. Define the first point of a vector used to move the member.
6. Define the second point of the vector.

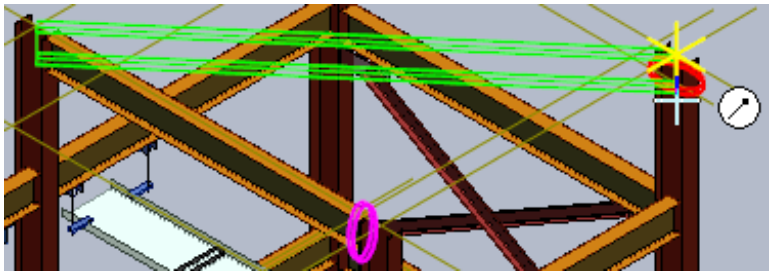
NOTE The frame connections might prevent you from moving the member as you would like. For example, a supported member has a seated frame connection to a supporting member. The seated frame connection prevents you from moving the supported member off of the supporting member. You can, however, slide the supported member along the supporting member as long as the seated frame connection is still valid. Consider copying and pasting a member that you want to move, and then deleting the original.

Move one end of a member

1. Click **Select** on the vertical toolbar.
2. Select **Frame Connections** in the **Locate Filter**.
3. Select the frame connection that is near the member end to move.



4. In the **Connection** option, select **By Rule**.
5. Define the new location for the member end.



Member System Prismatic Properties Dialog Box

Specifies the properties for the standard member or designed member that you are editing. For an explanation of the difference between a standard member and a designed member, see Members.

Member System Tab (Member System Prismatic Properties Dialog Box) (on page 813)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
Notes Tab (on page 817)

Member System Tab (Member System Prismatic Properties Dialog Box)

Specifies the properties for the member system.

Category

Select the properties to view for the member system.

Standard

Name

Displays the name of the member system. The member system name is based on the **Name Rule** selection. If you want to type a new name for the member system, in the **Name Rule** box, select **User Defined**, and then type a name for the member system in the **Name** box. If you change the **Type Category** of the member system after defining a custom name, the name resets to the default name for the new member type.

Name Rule

Specify the naming rule to use to name this member system.

MemberSystemTypeNameRule - Names the member system using this method: <member type>-<location>-<index number> where <member type> is the selected **Type** property, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Purlin-1-0003.

Default Name Rule - Names the member system using this method: MemberSystem-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberSystem-1-0045.

Unique Name Rule - Names the member system using this method: <parent system>-MemberSystem-<location>-<index number> where <parent system> is the name of the parent system that the Member System belongs to, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Structure System-MemberSystem-1-0001.

User Defined - Select this name rule to name the member system yourself using the **Name** box.

Parent System

Select the system to which the member system that you are placing belongs.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Continuity Type

Indicates the way in which the object system reacts when the system intersects another object system (your automatic splitting preference). The list is defined by the StructContinuity codelist.

Continuous indicates that the object system should split the other object system.

Intercostal indicates that the object system should be split by the other object system.

You cannot split objects that have a **Continuity Type** setting of **Continuous**.

Continuity Priority Number

Specifies the priority when two profile systems that have the same **Continuity** setting intersect. The profile system with the lowest continuity priority number penetrates the other system. For example, two profile systems intersect and the **Continuity** setting of each set to **Continuous**. The profile system with the lowest **Continuity Priority** number splits the other profile system. You can only modify this property at the root system.

Align

Indicates whether the software copies offsets from the frame connection at the member system end to the unsupported frame connection at the other member system end.

True indicates that Smart 3D copies the offsets.

False indicates the Smart 3D does not copy the offsets.

Start East / **Start X**

Displays the X-coordinates of the start of the member relative to the active coordinate system.

Start North / **Start Y**

Displays the Y-coordinates of the start of the member relative to the active coordinate system.

Start Elevation / **Start Z**

Displays the Z-coordinates of the start of the member relative to the active coordinate system.

End East / **End X**

Displays the X-coordinates of the end of the member relative to the active coordinate system.

End North / **End Y**

Displays the Y-coordinates of the end of the member relative to the active coordinate system.

End Elevation / **End Z**

Displays the Z-coordinates of the end of the member relative to the active coordinate system.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.
- For plate and profile system part properties, parent systems are shown.
- For assembly connection properties, all connected objects are shown.
- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.
- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group with which the selected objects are currently

associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.

New location

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group

Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

NOTE We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Member Part Prismatic Properties Dialog Box

Specifies the properties for the member part of the standard member that you are editing. For an explanation of the difference between a member system and a member part, see Members.

Member Part Tab (Member Part Prismatic Properties Dialog Box) (on page 818)
Cross Section Tab (Member Part Prismatic Properties Dialog Box) (on page 824)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
Notes Tab (on page 817)

Member Part Tab (Member Part Prismatic Properties Dialog Box)

Specifies the properties for the member.

Category

Select the properties to view for the member. Member part properties are divided into several different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, **Surface Treatment and Coating**, **Responsibility**, and **End Releases**. You select which category that you want to define values for by using the **Category** option.

Standard

Name

Displays the name of the member part. The member part name is based on the **Name Rule** selection. If you want to type a new name for the member part, in the **Name Rule** box, select **User Defined**, and then type a name for the member part in the **Name** box.

Name Rule

Specify the naming rule to use to name this member part.

- **Member Part Type Name Rule** - Names the member part using this method: <member type>-<location>-<index number> where <member type> is the selected **Type** property, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Beam-1-0003.
- **Default Name Rule** - Names the member part using this method: MemberPartPrismatic-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberPartPrismatic-1-0045.
- **Unique Name Rule** - Names the member part using this method: MemberPartPrismatic-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberPartPrismatic-1-0045.
- **User Defined** - Select this name rule to name the member system yourself using the **Name** box.

NOTE If you plan to export this model to Tekla Structures using CIMsteel, you must know that Tekla limits part names to 21 characters. If you use a Smart 3D naming rule that creates a member part name that is longer than 21 characters, Tekla truncates the name during the CIMSteel import into Tekla. This truncation can cause all your member parts to have the same name in Tekla because Smart 3D naming rules have the unique index at the end of the name.

Parent System

Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Length

Displays the length of the member without cutbacks applied. You cannot change this value.

Cut Length

Specifies the length of the member with cutbacks applied. You cannot change this value. A cutback is that part of a member removed by an assembly connection or by a manually

placed trim (a cope, for example).

Reporting Requirements

Specify whether this object is reported.

Reporting Type

Select the reporting requirements code for the object. Valid codes are defined in Catalog in the **Reporting Type** select list.

Piece Mark

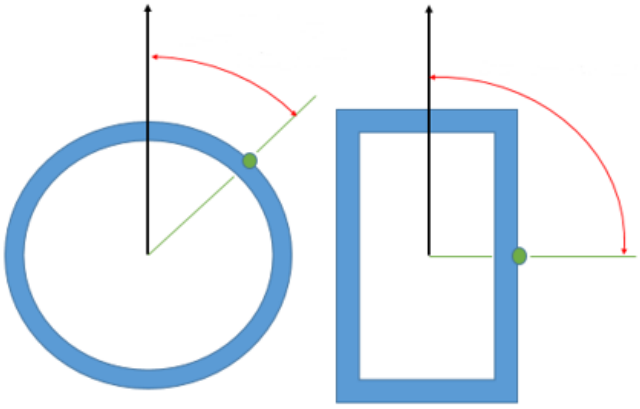
Specifies the piece mark of the member. Piece marks are typically used when importing and exporting member through CIMsteel. If a piece mark displays, the mark is usually the identity of the manufactured part from the other software package.

Assembly Mark

Specifies the assembly mark of the member. If the member was imported through CIMsteel, the assembly mark is usually the identity of the manufactured assembly to which this member belongs.

Seam Angle

Defines the location of the seam. This angle is measured clockwise from orientation vector of the member looking down the member from the start end.



Seam Angle is only available on cross-sections that use the **Has Seam** definition. Refer to the *Smart 3D Symbol 2D User's Guide* and *Symbol 2D Reference Data Guide* for more information about cross-sections.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

End Releases

Start Member Release

Select the directions to release at the start of the member part. Directions are defined in the local coordinate system of the member system. If the combination of directions is not available, select the **User Defined** option and define the releases yourself.

Start X Displacement

Defines if the X direction at the start of the member part is fixed or free.

Start Y Displacement

Defines if the Y direction at the start of the member part is fixed or free.

Start Z Displacement

Defines if the Z direction at the start of the member part is fixed or free.

Start X Rotation

Defines if the X moment direction at the start of the member part is fixed or free.

Start Y Rotation

Defines if the Y moment direction at the start of the member part is fixed or free.

Start Z Rotation

Defines if the Z moment direction at the start of the member part is fixed or free.

End Member Release

Select the directions to release at the end of the member part. Directions are defined in the local coordinate system of the member system. If the combination of directions is not available, select the **User Defined** option and define the releases yourself.

End X Displacement

Defines if the X direction at the end of the member part is fixed or free.

End Y Displacement

Defines if the Y direction at the end of the member part is fixed or free.

End Z Displacement

Defines if the Z direction at the end of the member part is fixed or free.

End X Rotation

Defines if the X moment direction at the end of the member part is fixed or free.

End Y Rotation

Defines if the Y moment direction at the end of the member part is fixed or free.

End Z Rotation

Defines if the Z moment direction at the end of the member part is fixed or free.

See Also

Member Part Prismatic Properties Dialog Box (on page 818)

Cross Section Tab (Member Part Prismatic Properties Dialog Box)

Specifies the properties for the cross-section of a standard member part.

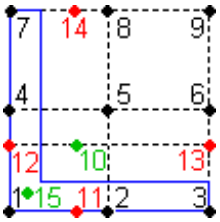
Section Standard

Specifies the section library from which you selected the section name. Sections are defined in the reference data.

Member Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

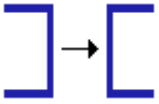


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle by which the section is rotated about the member axis.

Reflect



Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

Material

Select a material for the member. Materials are defined in the **AllCommon.xls** workbook or in the Catalog task.

Grade

Select a material grade for the member. Material grades are defined in the **AllCommon.xls** workbook or in the Catalog task.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit

this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the

corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric About Y

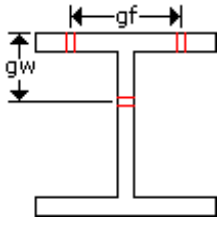
Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Flange Gage (gf)

Displays the bolt gage for the flange. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Gage (gw)

Displays the bolt gage for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.



Section Name

Specifies the section name. This name displays when you label members. This property is read-only. To edit this value, edit the cross-section's properties in Catalog or in the corresponding workbook.

Short Name

Displays the short name for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Alternate EDI Name

Displays the Electronic Data Interchange name for the section. This name is used when translating sections through CIMsteel. This property is currently not used.

Description

Specifies the description. This property is read-only. To edit this value, you need to edit the cross- section's properties in Catalog or in the corresponding workbook.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Unit Weight

Displays the weight of the section. The unit weight is defined in mass per length pound per foot (lbm/ft). This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Theoretical Maximum Yield Stress (Fy''')

Displays the maximum yield stress for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Statical Moment at Point in Flange (Qf)

Displays the first moment of area for the flange. This property is read- only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Statical Moment at Mid Depth of Section (Qw)

Displays the first moment of area for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Normalized Warping Function (Wno)

Displays the normalized warping function. The function is defined in square inches or square millimeters. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Beam Buckling Factor (X1)

Displays the beam buckling factor. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Beam Buckling Factor (X2)

Displays the beam buckling factor. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Thickness (tw)

Displays the web thickness for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Depth (d)

Displays the web depth. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Flange Thickness (tf)

Displays the flange thickness for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Flange Width (bf)

Displays the width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Distance to Web Toe Fillet (kdetail)

Displays the distance from the outer face of the flange to the web toe of the fillet of the rolled shape or the equivalent distance on welded section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Distance to Web Toe Fillet (kdesign)

Displays the distance from the outer face of the flange to the web toe of the fillet of the rolled shape or the equivalent distance on the welded section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Group Id

Displays the material group identification for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Designed Member Properties Dialog Box

Specifies the properties for the designed member that you are editing. For an explanation of the difference between a standard member and a designed member, see Members.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

Occurrence Tab (Designed Member Properties Dialog Box) (on page 829)
Cross Section Tab (Designed Member Properties Dialog Box) (on page 832)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
Notes Tab (on page 817)

Occurrence Tab (Designed Member Properties Dialog Box)

Specifies the properties for the select designed member.

Category

Select the properties that you want to view for the member. Only **Standard** properties are available for designed members.

Show Dimensional Legend

Displays a graphic indicating the different dimensions that can be specified for the selected designed member.

Standard

Name

Displays the name of the designed member. The designed member name is based on the **Name Rule** selection. To type a new name for the designed member, select **User Defined** in the **Name Rule** box, and then type a name for the designed member in the **Name** box.

Name Rule

Specify the naming rule to use to name this designed member. You can select one of the listed rules or select **User Defined** and type the designed member name in the **Name** box.

Parent System

Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.

Section Standard

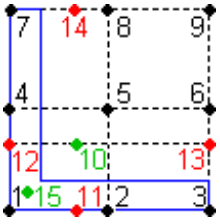
Specifies the section library from which you selected the section name. Sections are defined in the reference data.

Member Section Name

Defines the cross-section for the member. If you know the section name that you want, you can type it. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type BUIH* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. For more information about reference data, see *Structure Reference Data Guide*.

Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the



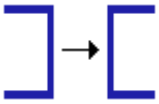
section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle by which the section is rotated about the member axis.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option is when you want the flanges of a channel section to point in the opposite direction.



Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric about X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric about Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Cross Section Tab (Designed Member Properties Dialog Box)

Specifies the properties for the cross-section of a designed member.

Name

Displays the cross-section name for the designed member. Sections are defined in the reference data. For more information about reference data, see *Structure Reference Data Guide*.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only.

To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric About Y

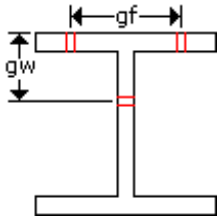
Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Flange Gage (gf)

Displays the bolt gage for the flange. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Gage (gw)

Displays the bolt gage for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.



Plastic Neutral Axis (xp)

Displays the distance from the designated edge of the cross-section's plastic neutral axis (PNA) along the local x-axis.

Plastic Neutral Axis (yp)

Displays the distance from the designated edge of the cross-section's plastic neutral axis (PNA) along the local y-axis.

Section Name

Specifies the section name. This name displays when you label members. This property is read-only. To edit this value, edit the cross-section's properties in Catalog or in the corresponding workbook.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to

edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Unit Weight

Displays the weight of the section. The unit weight is defined in mass per length pound per foot (lbm/ft). This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Nonlinear Plate System Properties Dialog Box

Specifies the properties for the root or leaf plate system of the designed member that you are editing. A designed member contains multiple plate systems that you can individually edit.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Nonlinear Plate System Properties Dialog Box) (on page 187)
Material Tab (Nonlinear Plate System Properties Dialog Box) (on page 190)
Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box - Designed Member) (on page 840)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
General Tab (on page 125)

Main Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the general properties of the nonlinear extruded plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type

(Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.

- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the

property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

Specifies the level of tightness as it applies to the entire plate system.

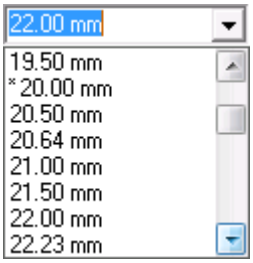
Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on

page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the material properties for the nonlinear extruded plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

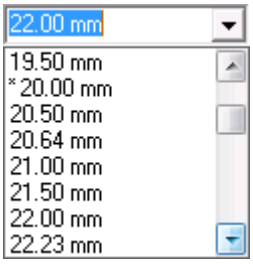
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box - Designed Member)

Specifies the plate orientation rules for the plate system of the designed member that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. A plate on a designed member can be **Flange Plate**, **Web Plate**, or **Tube Plate**. Plate system types are defined by the designed member cross-section. You cannot change the type.

Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



General Tab

The **General** tab displays the properties that were selected by you or automatically determined by the software at creation. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

The properties displayed duplicate those available on other tabs, such as the **Main** tab. You can edit the properties on these tabs.

Plate Part Properties Dialog Box

Specifies the properties for the plate part that you are editing.

Main Tab (Plate Part Properties Dialog Box) (on page 125)
Material Tab (Plate Part Properties Dialog Box) (on page 127)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
General Tab (Plate Part Properties Dialog Box) (on page 127)
Weight & CG Tab (on page 129)
Extended User Attributes Tab (on page 129)
Routing Tab (on page 129)

Main Tab (Plate Part Properties Dialog Box)

Specifies the general properties of the plate part.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the object.

PlatePartRule - Creates a name based on the parent root plate system name, the location within the parent root plate system, and the parent assembly name.

User Defined - Name the plate part yourself by using the appropriate box.

Type

Displays the type of the plate part. It is inherited from the parent system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Displays the parent system for the object.

Surface Geometry Type

Displays the Molded Forms command used to create the root parent plate system. It is inherited from the parent system.

Specification

Specifies the specification for the object. This property is inherited from the parent system.

Description

Specifies a description for the object.

Tightness

Specifies the water tightness of the object. It is inherited from the parent system.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Material Tab (Plate Part Properties Dialog Box)

Displays the material properties for the plate part that you are editing. The material properties are inherited from the parent system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Actual Thickness

Displays the material thickness for the object.

General Tab (Plate Part Properties Dialog Box)

Specifies general properties of the plate part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Category: Standard

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Plate Part Properties Dialog Box)* (on page 125).

PlateTightness

Specifies the water tightness of the plate part. The list is defined by the StructPlateTightness codelist.

PlateType

Specifies the type of the plate part. The list is defined by the StructPlateType codelist.

Thickening Technique Requested

Specifies the technique requested to thicken the plate part.

This property affects only the behavior of plate parts derived from systems. The property defaults to **Procedural** at creation. Changing the value of this property can affect the geometry of the part as described below.

Standalone plate parts initially display nothing for this property. You can change the value, but those changes do not change the part or its geometry. Smart 3D thickens all standalone plate parts, including collars, using procedural thickening.

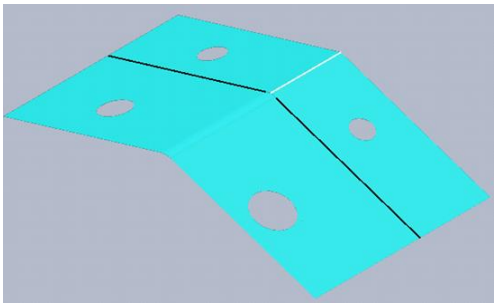
Undefined (blank) indicates that Smart 3D selects the appropriate technique. Smart 3D automatically changes this value to **Procedural**.

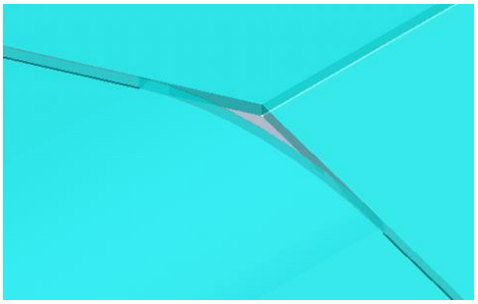
Procedural indicates the part is thickened using the standard thickening method provided by the ACIS modeler. This works for the vast majority of parts, and you should not change this value unless there is a specific reason to try a different technique.

NOTE If the thickening fails using ACIS, and the part is a hull part, the software automatically tries the **NonProcedural** thickening technique. This option only controls which method the software tries first. You can determine which method Smart 3D actually used from the **Thickening Technique Used** property.

Procedural with slit is used for parts that have internal slits or discontinuities in the surface of the system that form part of the boundary of parts on the system. The currently-supported example of this is a surface with a combination of split and bent knuckles joined end to end. Because Smart 3D does not support splitting knuckles and changing the properties of different segments, these surface must be constructed with the correct geometry to reflect the bent and split portions. Parts that are bounded by the slit, and potentially parts on the system that are within half a meter of the slit, must have this value set to assure a proper trim. Setting this attribute on parts that do not need it should not cause a failure but may cause the part to take more time to detail. The following pictures show an example of this type of surface, first the full surface and then a close-up in the region of the slit or discontinuity.

The plate system in these pictures is split into three parts by the knuckle (shown as white) and the two black seams. You must be sure that a single part does not touch both sides of the slit.





NonProcedural is an infrequently used option. Historically, a much higher percentage of parts failed to thicken with the ACIS modeler, and many designers used a specific hull form definition tool that produced a surface with very specific characteristics. **NonProcedural** thickening provided a tool that would thicken almost any part on this type of surface, even if the ACIS modeler failed. The ACIS modeler has improved greatly, and the **NonProcedural** thickening method has not been enhanced to handle the surfaces produced by many of the hull form tools currently in use. Setting this option may solve a failure-to-thicken problem on a hull part.

NOTE If **NonProcedural** thickening fails for a hull shell part, Smart 3D automatically tries to thicken the part using **Procedural** thickening. This option only controls which method the software tries first. You can determine which method Smart 3D actually used from the **Thickening Technique Used** property.

Thickening Technique Used

Displays the technique used to thicken the plate part. This is blank for a standalone plate part, although **Procedural** is always used for these parts. It displays one of the values defined for **Thickening Technique Requested** for a system-derived part. The meaning for these values is the same as defined for **Thickening Technique Requested**.

Category: Planning

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

Weight & CG Tab

Displays the center-of-gravity and the weight of the part.

Type

Displays the type of weight: **Dry Weight** or **Wet Weight**.

Weight

Displays the weight of the plate part pertaining to the weight type.

CoG X

Displays the position of center of gravity along the X-axis in the **CoG Coordinate System**.

CoG Y

Displays the position of center of gravity along the Y-axis in the **CoG Coordinate System**.

CoG Z

Displays the position of center of gravity along the Z-axis in the **CoG Coordinate System**.

CoG Coordinate System

Specifies the coordinate system to use to measure the center of gravity.

Extended User Attributes Tab

Displays the user attributes that have been bulkloaded on the report data object related to the part. Changes to these attributes do not cause the related manufacturing part to go out-of-date. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Routing Tab

Specifies routing properties for the part. By default, no routing properties are set.

Create

Sets routing properties for the part.

Delete

Deletes the routing properties for the part.

Rule

Specifies a predefined rule set or a user-defined rule that is applicable to the part.

Workcenter

Select an assembly workcenter where the part is produced from a facility-defined rule set. The list contains all available workcenters regardless of their level in the workcenter hierarchy. Examples of workcenters include docks, shops, bays, and panel lines. Workcenters may also be areas and zones where a block is assembled from multiple assemblies.

Stage Code

Specifies the workcenter code.

No. Of Actions

Specifies the number of actions required to produce the part.

Action

Specifies the name of the action.

Machine

Specifies the type of machine used to perform the required operation.

Code

Specifies the machine code.

Frame Connection Properties Dialog Box

Specifies the properties for the frame connection that you are editing.

See Also

General Tab (Frame Connection Properties Dialog Box) (on page 850)
Edit a Frame Connection (on page 809)
Relationship Tab (on page 123)
Notes Tab (on page 817)

General Tab (Frame Connection Properties Dialog Box)

Specifies the properties for the frame connection.

TIP The *supported* member is the member that you are placing. The *supporting* member is the existing member in the model to which you are connecting.

Category

Select the type of properties to view for the selected frame connection.

Type

Displays the type of frame connection. This box is read-only. See Frame Connections for more information about the different types that are available.

Property

Displays all properties associated with the selected frame connection. The list of available properties depends on what was defined in the reference data for the frame connection type.

Value

Specifies the values for the frame connection properties.

Name

Specifies the frame connection name.

Name rule

Specifies the naming rule used to name the frame connection.

- **Default Name Rule** - Names the frame connection using the format "<frame connection type>-<location>-<index>" where <frame connection type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, Axis-Along-1-0043.

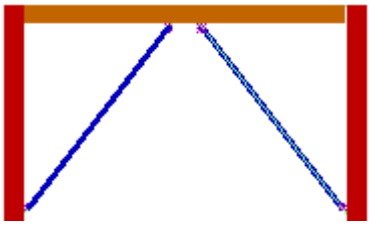
- **User Defined** - Select this rule to name the frame connection yourself using the **Name** box.

Position rule

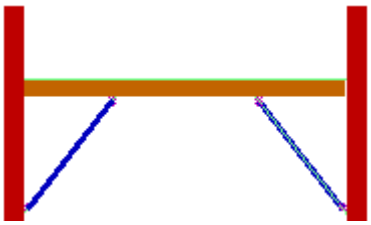
Defines how the frame connection is to behave when the supporting member is moved. You can select one of three options:

- **Intersection** - The member system lengthens or shortens to maintain the connections with the supporting member. The end of the supported member system slides to a new location on the supporting member.
- **Ratio** - The member system lengthens or shortens to maintain the connection with the supporting member. The end of the supported member system stays in the same relative position (that you can define) along the supporting member system. This option is similar to the **Distance** option except that you define a percentage ratio from the supporting member end.
- **Distance** - The member system lengthens or shortens to maintain the connection with the supporting member. The end of the supported member system stays in the same position (that you can define) along the supporting member system. This option is similar to the **Ratio** option except that you define an actual distance from the supporting member end.

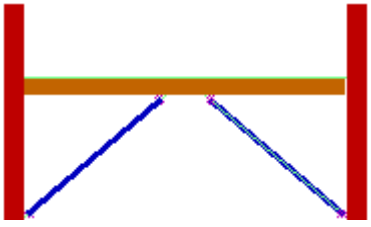
Original Position:



Intersection Position Rule:



Ratio and Distance Position Rule:



Distance along

Type the distance from the supporting member end that the supported member is positioned. You must include the units of measurement when defining this distance. The supporting member end that is measured from, is defined using the **End** option. This option is available

when you set **Position Rule** to **Distance**.

Ratio

Type the ratio of the supporting member length that the supported member is positioned. For example, type **.25** if you want the supported member a fourth of the way along the supporting member. Type **.333** if you want the supported member a third of the way, and so forth. Which supporting member end that is measured from is defined using the **End** option. This option is available when you set **Position Rule** to **Ratio**.

End

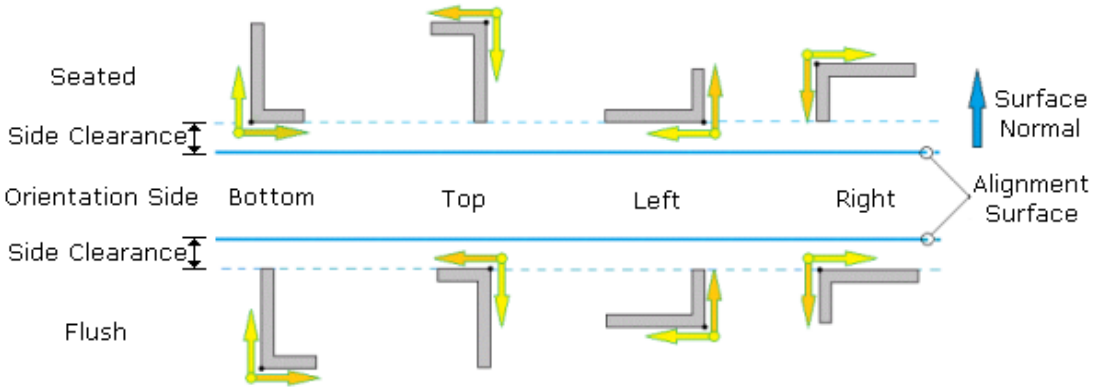
Specifies which end of the supporting member that the ratio or the distance-along distance is measured. You can select **Start**, **End**, or **Auto**. Start is the first member end that was placed. End is the second member end that was placed. If you select **Auto**, the software automatically selects the supporting member end that is closest to the frame connection. The **Auto** setting is recommended so that you do not have to worry whether the supporting member was modeled left-to-right, right-to-left, top-to-bottom, or bottom-to-top.

Align Frame Connection Properties

Side Justification

Select the side of the plane on which to place the member.

- **None** - Aligns the member axis so that it is placed on the surface.
- **Seated** - Sets the member on top of the surface. The selected **Orientation Side** is nearest the surface.
- **Center** - The center of the cross-section is placed on the surface. The center is calculated by the top-bottom, left-right bounds of the cross-section shape.
- **Flush** - Uses the alignment surface's top and bottom extent to position the supported member. The supported member typically lies within the body of the alignment plane with one edge of the member flush with the alignment surface but can be offset.

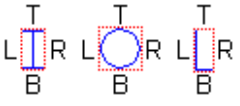


Orientation Control

If **On**, the member angle is controlled by the **Orientation Angle** value defined for the Align frame connection. If **Off**, you can set the angle of the member independently of the align surface by using the **Rotation** property for the member part.

Orientation Side

Specifies the side of the supported member's cross-section that is



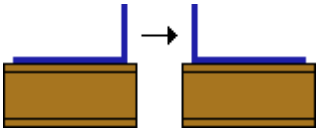
mated to the alignment surface. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.

Orientation Angle

When **Orientation Control** is **On**, this angle is added to the selected **Orientation Side** to rotate the member. If the Align frame connection is used on both ends of the member and you define a different angle value for each end, the angle defined at the start end of the member is used.

Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. An example of when to use this option is when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Extend Distance

Specifies the offset distance from the bounding surface along the member's axis.

Side Clearance

Specify the offset distance perpendicular to the align surface to place the member axis.

Lateral Distance

Specifies the offset distance in the plane of the align surface to place the member axis.

Axis Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction after the two cardinal points are aligned.

Y Offset

Specifies an offset to apply in the y-direction after the two cardinal points are aligned.

Z Offset

Specifies an offset to apply in the z-direction after the two cardinal points are aligned.

Coordinate System

Specifies the coordinate system to use for the offset values. Local is the local coordinate system of the supporting member.

Supporting CP

Specifies to which cardinal point on the supporting member system to align the supported member system's cardinal point. You can specify any cardinal point number, or select 0 to use the cardinal point with which the supporting member was placed.

Gap Frame Connection Properties

There are three members in a gap frame connection:

- The *target* member is the member always to move and is the owner of the frame connection. The target member is shown as blue in the figure below.
- The *primary* member is the member to which the other two members are attached. The primary member is shown as red in the figure below.

- The *secondary* member is the third member in the joint. The software does not require the secondary member to be in the same plane as the target member. The secondary member is shown as orange in the figure below.

Actual Gap Distance (G)

Displays the actual gap distance between the target and secondary member. This distance is shown as G in the figures below.

Actual Centerline Distance (X)

Displays the actual gap between the center line of the target member and the center line of the secondary member. This distance is shown as X in the figures below.

Actual Overlap Distance (L)

Displays the distance between the lowest point of the target and secondary member saddle on the primary member to the point where the target and secondary member intersect. This distance is shown as L in the figures below.

Actual Overlap Distance (A)

Displays the amount of overlap between the target and secondary member. This distance is shown as A in the figures below.

Allowable Centerline Distance

Displays the calculation of the primary member diameter times the **Centerline Design Factor**. The centerline distance should be less than or equal to this value.

Allowable Overlap Distance (Fmin × B)

Specify the minimum overlap distance (A) that is acceptable.

Allowable Overlap Distance (Fmax × B)

Specify the maximum overlap distance (A) that is acceptable.

Design Passed (Centerline)

Display **True** if the centerline design passed. Displays **False** if the centerline failed. The centerline design pass/fail is calculated by comparing actual centerline distance with the allowable centerline distance.

Design Passed (Overlap)

Display **True** if the overlap design passed. Displays **False** if the overlap failed. The overlap design pass/fail is calculated by (Min. Factor × B) < A < (Max. Factor × B) and then comparing that value to the **Allowable Overlap Distance (Fmin × B)** and the **Allowable Overlap Distance (Fmax × B)** values. You define the minimum and maximum design factors using the **Overlap Design Factor (Fmin)** and **Overlap Design Factor (Fmax)** properties. The B distance is shown in the figures below.

Direction

Select the direction to calculate the gap.

- **Axially Along Surface** - Select this mode when you want to define the offset as the distance between the center lines of the target and secondary members. This distance is measured parallel to the primary member centerline.
- **Axially and Flush Along Surface** - **This option is currently not available.** Select this mode when you want to define the offset as the distance between the center lines of the

target and secondary members and you want the target member to be flush with the outside of the primary member. The axially distance is measured parallel to the primary member centerline. Use the **Flush Direction** and **Flush Offset** options to refine exact flush placement.

- **Radially Along Surface** - Select this mode when you want to define the offset as the distance between the closest points on the hull of the target and secondary members. This distance is measured around the hull of the primary member.

Offset Required

Type the gap distance that you want between the target and secondary member. Select how you want to define this distance using the **Offset Type** box.

Centerline Distance Roundoff

Type the distance to which the centerline gap should be rounded off.

Centerline Design Factor

Type the ratio of the primary member section size to use in calculating whether the centerline design passed or failed. The default value is **0.25**, which specifies that the primary member diameter be divided by 4 (D/4).

Flush Direction

This option is currently not available. Specify which direction you want the target member to move to become flush with the primary member.

Flush Offset

This option is currently not available. Specify the distance from the primary member hull edge to the target hull edge. An offset of zero indicates that the two member edges are perfectly flush.

Offset Along

Defines if the target member moves along the primary member or along the secondary member to create the gap. This property is only available for Gap Single Axial gap connections.

Offset Type

Select how you want to specify the **Offset Required** value.

- **Gap** - The **Offset Required** distance is between the two closest points of the target member hull and the secondary member hull. This is shown as G in the figures below.
- **Centerline** - The **Offset Required** distance is between the center line of the target member and the center line of the secondary member. This is shown as X in the figures below.
- **Overlap** - The **Offset Required** distance is between the lowest point of the target and secondary member saddle on the support member to the point where the target and secondary member intersect. This is shown as L in the figures below.

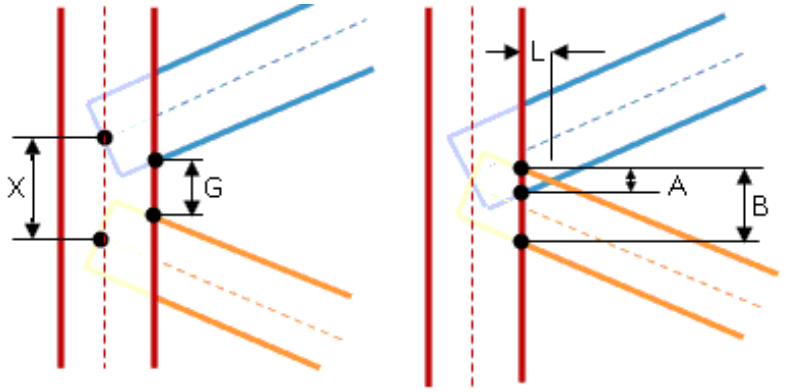
Overlap Design Factor (Fmin)

Defines the minimum design factor for the calculation: Min $\times$ B < A < Max $\times$ B. The default value is 0.3. This calculation is used to determine if **Design Passed (Overlap)** passes or fails. The A and B distances are shown in the figures below.

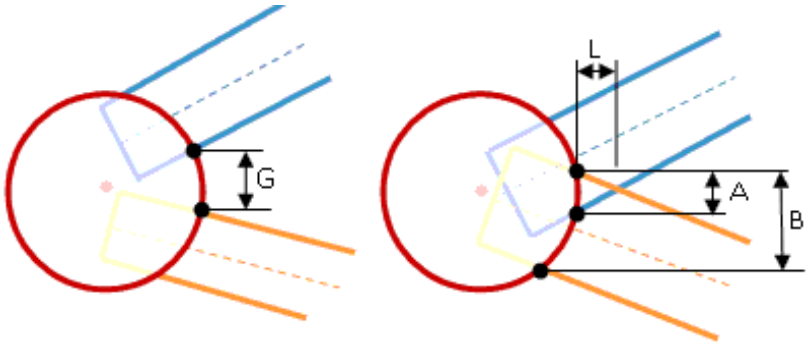
Overlap Design Factor (Fmax)

Defines the maximum design factor for the calculation: Min × B < A < Max × B. The default value is 0.5. This calculation is used to determine if **Design Passed (Overlap)** passes or fails. The A and B distances are shown in the figures below.

Axially Along Surface:



Radially Along Surface:



Seated, Flush, and Centerline Frame Connection Properties

Side

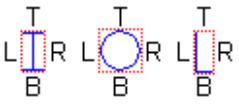
Select the side of the supporting member on which you want to place the supported member.

Offset

Specify the distance to place the supported member from the supporting member. For seated and flush frame connections, the offset is between the side of the supporting member that you specified with the **Side** option and the supported member's side that you specify with the **Edge** option. For centerline frame connections, the offset is between centerline of the supporting member and the supported member's side that you specify with the **Edge** option.

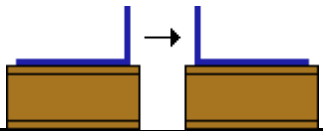
Edge

Specifies the side of the supported member's cross-section that is mated to the supporting member. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.



Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. For



example, when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.

Surface Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction.

Y Offset

Specifies an offset to apply in the y-direction.

Z Offset

Specifies an offset to apply in the z-direction.

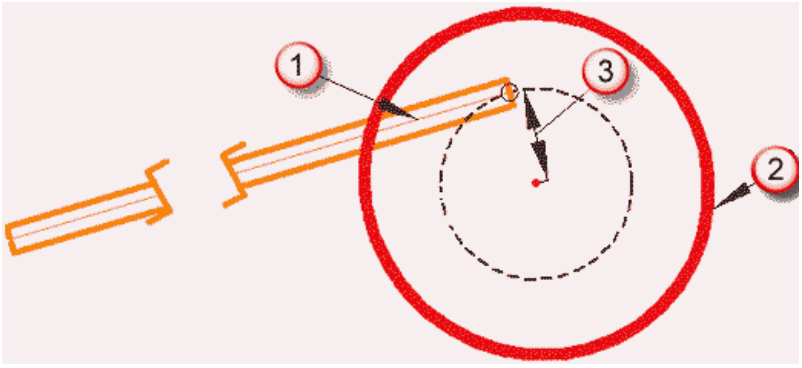
Coordinate System

Specifies the coordinate system to use for the offset values.

Tangent Frame Connection Properties

Offset Distance

Type the distance (**3**) from the centerline of the supporting member (**2**) to the tangent connection point. This distance is the radius of the circle to which the supported member (**1**) is made tangent.



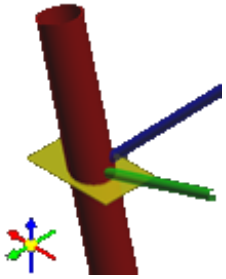
Side

Select **Left** or **Right** to define on which side of the supporting member the supported member is placed. To determine left and right, image you are standing on the start end of the member with your head towards positive global Z looking down the length of the member. Your left and right determine the direction of **Left** and **Right**.

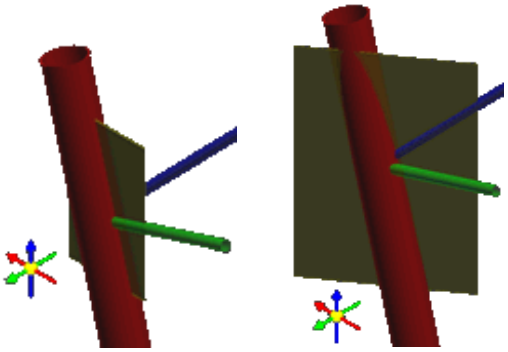
Tangent Plane

Controls the plane on which the tangent circle is drawn. You can select:

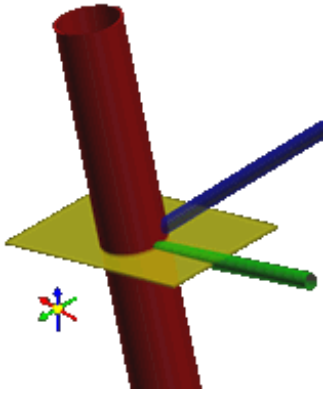
- **Horizontal** - The tangent circle is parallel to the X-Y plane (normal to global Z) regardless of the slope of any of the members. For example, you are creating a tangent connection between the sloping green beam and the sloping red column, or between the sloping blue beam and the sloping red column.



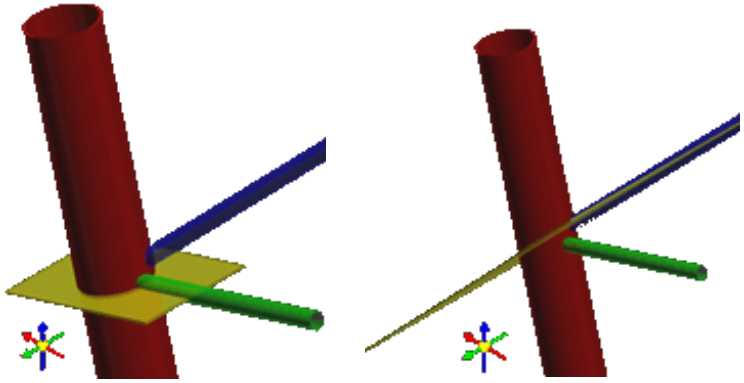
- **Vertical** - The tangent circle is parallel to global Z and is aligned with the centerline of the supported member (green member in the left figure, blue member in the right figure).



- **Normal to Supporting** - The tangent circle is normal to the supporting member (the sloping red column).



- **In Plane of Supported** - The tangent circle is parallel to the centerline of the supported member and normal to the supporting member centerline (green member in the left figure, blue member in the right figure).



Vertical Corner Brace Frame Connection Properties

X Offset

Specifies the offset to apply in the x-direction.

Y Offset

Specifies the offset to apply in the y-direction.

Z Offset

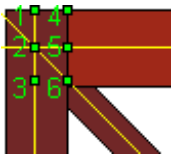
Specifies the offset to apply in the z-direction.

Coordinate System

Select whether the offset values are defined relative to the global coordinate system or the member's local coordinate system.

Work Point

Specifies the work point location. There are six work point locations that you can select.



1 - Primary Center - Secondary Far Side
2 - Primary Center - Secondary Center
3 - Primary Center - Secondary Near Side
4 - Primary Near Side - Secondary Far Side
5 - Primary Near Side - Secondary Center
6 - Primary Near Side - Secondary Near Side

See Also

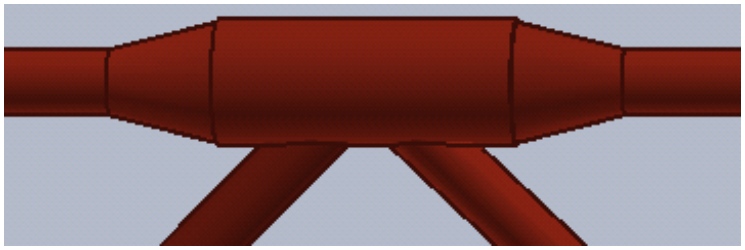
Frame Connection Properties Dialog Box (on page 850)

SECTION 8

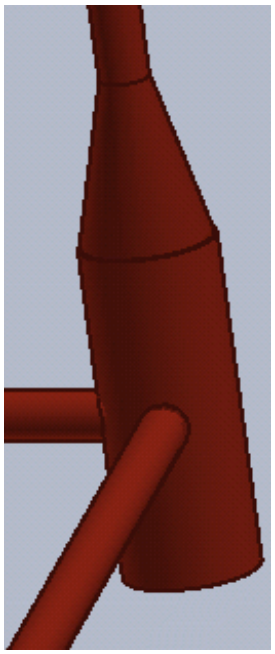
Place Can

Places a can in the model. A can is a reinforcing connection piece on a member to increase the surface area available for connections or to provide increased wall thickness or both. You can place cans anywhere along a linear member (the support leg) including the end of the member. There are three can types:

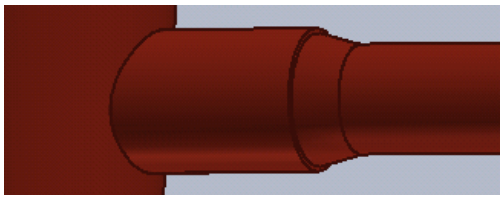
- Inline can - This can is placed anywhere along the length of a support leg. The support leg is split into two parts and the can is placed between those parts.



- End can - This can is placed at the end of the support leg. As a general rule, the end of the support leg does not frame into another member.



- Stub-end can - This can is also placed at the end of the support leg, which generally does frame into another member.



The software calculates the can length based on any transverse members that you select and the L2 and L3 parameters that you define.

Can Limitations

- SmartSketch precision placement commands such as **Point Along** and **PinPoint** are not available when placing an in-line can. The in-line can's location is determined by where you select the supporting member or by the location of any transverse members assigned to the can.
- Stub-in and end cans locations are determined by the frame or split connection that you select during placement.
- The linear member that you select to be the supporting leg must be a designed member with a BUTube cross-section. You cannot select curved members for the supporting leg or for the transverse members.

Place Can Ribbon

Can Properties

Activates the **Can Properties** dialog box. Use this dialog box to specify additional properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Can Properties Dialog Box* (on page 872).

Select Primary Member

Select the support leg on which to place the can. The support leg is split at the location you specify and highlights in red during can placement. If you are placing an inline can, you can select a BUTube designed member. You cannot select a curved member as the primary member. If you are placing an end can, you must select an unsupported frame connection. If you are placing a stub-end can, you can select any split connection or frame connection except unsupported frame connections.

If you are modifying an existing can, this option is not available.

Select Member Part

Select the member part on which to place the stub-end can. This option is only available with the **Can Type** is set to a stub-end can and you have selected a split connection for the can's location.

Select Transverse Members

Select the transverse members with which you want to base the can length. All transverse members that are related to the can highlight in blue. You can select any member systems, except curved members, that are connected to the primary member. You can use **Cancel** ✕ to clear selected transverse members and **Accept** ✓ to confirm the transverse members that you selected are the ones you want to use.

If you are modifying more than one existing can, this option is not available.

Finish

Click to place the can in the model.

Cancel

Rejects the transverse members that you have selected.

Accept

Confirms the transverse members that you have selected.

Primary Member Information

Activates a small dialog box that displays the name, outside diameter, inside diameter, and plate thickness of the selected primary member. This information is useful when selecting an appropriate can type from the Catalog or when defining custom can specifications.

If you are modifying an existing can, this dialog box displays the name, outside diameter, inside diameter, and plate thickness of the support legs on either side of the selected can.

Can Type

Specifies the type of can to place: stub-end, inline, or end can. When you select a can type, the defaults settings from the Catalog are loaded.

Tube / Cone Parameters

Click to swap between defining cone parameters and defining tube parameters.

Tube Parameters

Outer/Inner Diameter

Specifies the can outer diameter or the can inner diameter depending on the can type you have selected.

- If you selected a pre-defined can from the Catalog, the diameter value from the Catalog is displayed.
- If you selected a Custom Can-OD Match from the Catalog, the outer diameter of the selected primary member is displayed.
- If you selected a Custom Can-ID Match from the Catalog, the inner diameter of the selected primary member is displayed.
- If you selected a Custom Can-User Defined from the Catalog, you must type a diameter value in this box.

Can Thickness

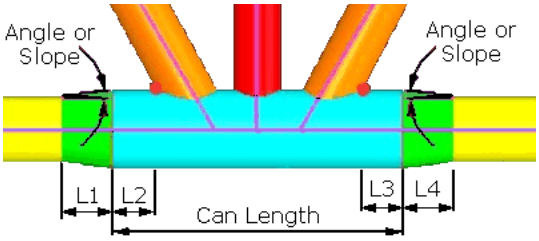
Select the plate thickness for the can plate.

Computation Method

Specifies how you want Smart 3D to measure the L2 and L3 minimum distances.

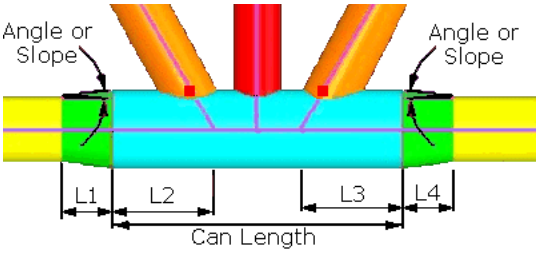
- If you select **Hull-Factor**, the software calculates the outside diameter of the can times the value you type in the **Factor** box. The result is used as the minimum L2 and L3 distances which are measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.

- If you select **Hull-Length**, the software uses the distance you specify in the **Length** box for L2 and L3, which are measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.



L2 and L3 Lengths when using a Hull Extension Method

- If you select **Centerline-Factor**, the software calculates the outside diameter of the can times the value you type in the **Factor** box. The result is used as the minimum L2 and L3 distances, which are measured from the corresponding end of the can to the centerline of the transverse member closest to that end of the can.
- If you select **Centerline-Length**, the software uses the distance you specify in the **Length** box for L2 and L3, which are measured from the corresponding end of the can to the centerline of the transverse member closest to that end of can.



L2 and L3 Lengths when using a Centerline Method

Factor

Specifies the factor to use in the extension method calculation. The default value is 0.5.

Length

Specifies the absolute distance to use in the extension method calculation. If you type a length that is shorter than the American Petroleum Institute's minimum allowable length, a message displays that one or more specified parameters are invalid.

Cone Parameters

Cones By

Specifies how you want to define the length of the cone transitions.

- If you select **Angle** the length of the transition is determined by the geometry of the can diameter, the support member cross-section, and the angle specified in the **Cone Angle** box. This is shown as **Angle** in the figures above.
- If you select **Length** the length of the transition is defined explicitly in the **Cone Length** box. This distance is shown as L1 and L4 in the figures above.
- If you select **Slope**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section size, and the slope value specified in

the **Cone Slope** box (the denominator of the slope). This is shown as **Slope** in the figures above.

NOTE During placement, both cone transitions are placed using one set of values. If you want the cone transitions on either end of the can to have different lengths or different angles, you can change these values independently on the *Can Properties Dialog Box* (on page 872) after placement.

Cone Angle

Specifies the angle of the cone transition. The length of the cone is adjusted so that the entered angle is obtained. The software does not create a cone if the angle you specify is less than 1 degree or greater than 89.0 degrees.

Cone Length

Specifies the length of the cone transition from the end of the support member to the start of the can. This distance is shown as **L1** and **L4** in the figures above.

Cone Slope

Specifies the slope of the cone transition. The length of the cone is adjusted so that the entered slope is obtained. The software does not create a cone if the slope you specify is greater than 100 (1/100) or less than 1^{-6}.

Cone Thickness

Select the plate thickness for the cone.

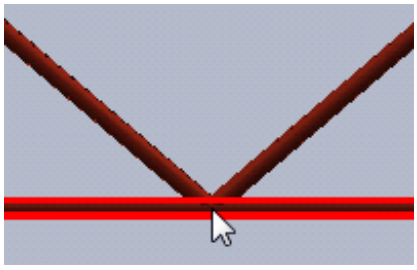
What do you want to do?

- *Place an in-line can* (on page 865)
- *Place an in-line can on cross bracing* (on page 866)
- *Place an end can* (on page 868)
- *Place a stub-end can* (on page 869)
- *Add a transverse member to a can* (on page 870)
- *Remove a transverse member from a can* (on page 871)

Place an in-line can

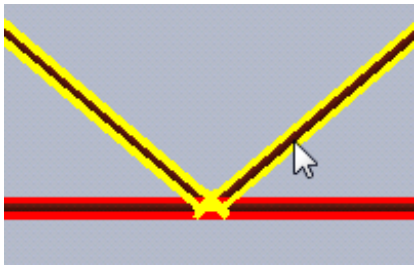
1. Click **Place Can** on the vertical ribbon.
2. From the Catalog browser or from **Can Type** on the ribbon, select the in-line can rule to place from the Catalog.

   TIP Select one of the Custom Inline Can types to change individual can parameters.
3. Select the location of the in-line can along the supporting leg member.

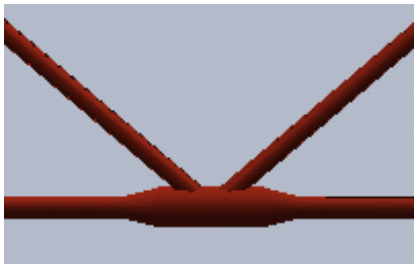


4. Click **Primary Member Information** on the ribbon.

   The software displays the name, outside diameter, inside diameter, and plate thickness of the selected supporting leg.
5. Select the transverse members that participate in the can. Transverse members are used in calculating the can length.

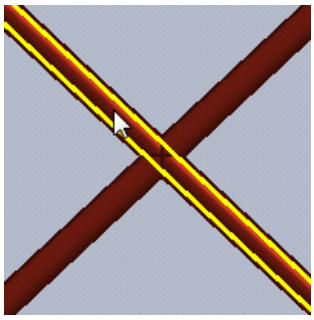


6. Click **Accept** ✓.
7. If you are placing a Custom Inline Can type, change the diameter, thickness, computation method, factor, and cone parameters on the ribbon to meet your needs. If you are placing a predefined can from the Catalog, these settings are read-only.
8. Click **Finish**.

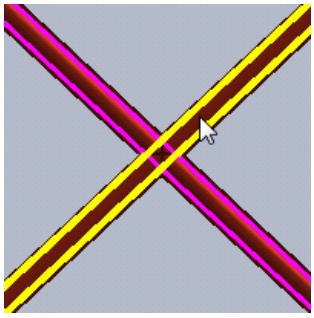


Place an in-line can on cross bracing

1. Click **Place Split** on the vertical toolbar.
2. In the **Place** box, select **By Object**.
3. In the **Split Status** box, select **Split First**.
4. Select the cross brace to split.



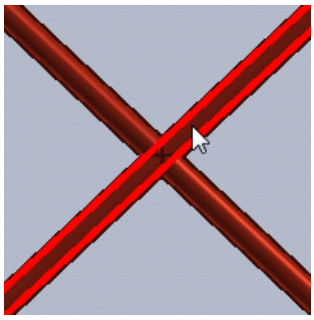
5. Click **Accept** ✓.
6. Select the other cross brace.



7. Click **Finish**.
8. Click **Place Can** on the vertical ribbon.
9. From the Catalog browser or from **Can Type** on the ribbon, select the in-line can rule to place from the Catalog.

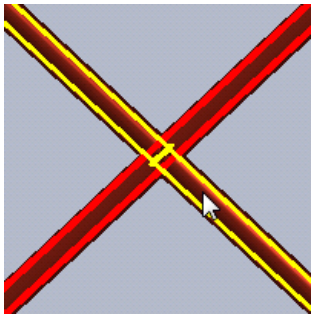
   TIP Select one of the Custom Inline Can types to change individual can parameters.

10. Select the cross brace you selected in step 6 (the cross brace that you did not split) near the intersection of the two cross braces.

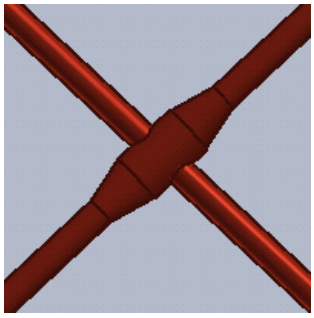


The software prompts you to select the transverse members.

11. Select the cross brace that you split (the cross brace you selected in step 4).



12. Click **Accept** ✓.
13. If you are placing a Custom Inline Can type, change the diameter, thickness, computation method, factor, and cone parameters on the ribbon to meet your needs. If you are placing a predefined can from the Catalog, these settings are read-only.
14. Click **Finish**.

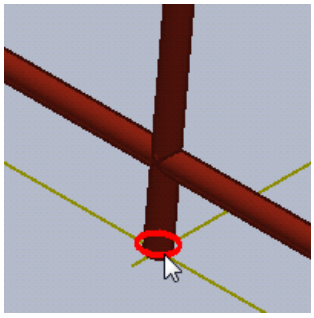


Place an end can

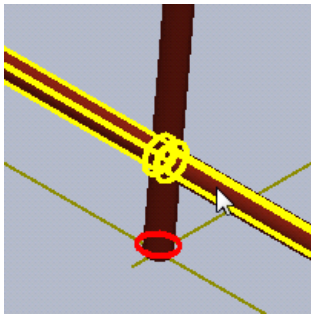
1. Click **Place Can** on the vertical ribbon.
2. From the Catalog browser or from **Can Type** on the ribbon, select the end can rule to place from the Catalog.

   TIP Select one of the Custom End Can types to change individual can parameters.
3. Select the unsupported frame connection at the member end where you want the end can.

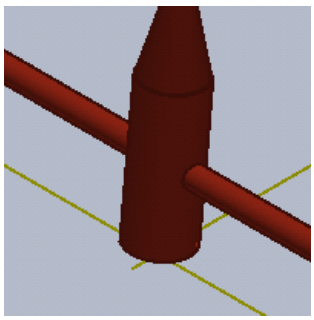
   TIP End cans are only placed on unsupported frame connections.

   
4. Click **Primary Member Information** on the ribbon.

   The software displays the name, outside diameter, inside diameter, and plate thickness of the selected supporting member.
5. Select the transverse members, if any, that participate in the can. Transverse members are used in calculating the length of the can.

   
6. Click **Accept** ✓.
7. If you are placing a Custom End Can type, change the diameter, thickness, computation method, factor, and cone settings on the ribbon to meet your needs. If you are placing a predefined can from the Catalog, these settings are read-only.

8. Click **Finish**.



Place a stub-end can

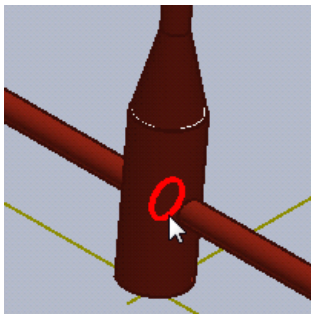
1. Click **Place Can** on the vertical ribbon.
2. From the Catalog browser or from **Can Type** on the ribbon, select the stub-end can rule to place from the Catalog.

   TIP Select one of the Custom Stub End Can types to change individual can parameters.
3. Select the frame connection at the member end where you want the stub-end can.

   TIP Stub-end cans cannot be placed on unsupported frame connections.

   or

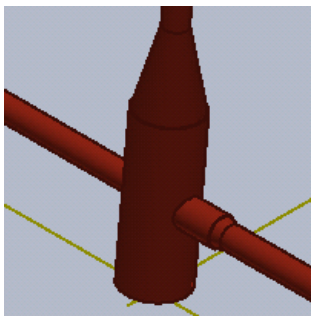
   Select the split connection where you want to place the stub-end can.



4. If placing the stub-end can at a split connection, select the member part on which to place the stub-end can.
5. Click **Primary Member Information** on the ribbon.

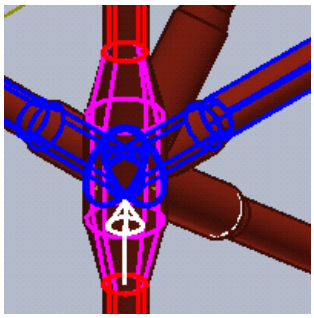
   The software displays the name, outside diameter, inside diameter, and plate thickness of the selected supporting member.
6. Select the transverse members, if any, that participate in the can. Transverse members are used in calculating the can length.
7. Click **Accept** ✓.
8. If you are placing a Custom Stub End Can type, change the diameter, thickness, computation method, factor, and cone settings on the ribbon to meet your needs. If you are placing a predefined can from the Catalog, these settings are read-only.

9. Click **Finish**.

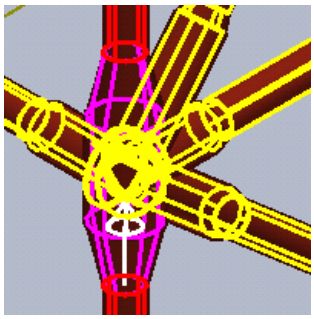


Add a transverse member to a can

1. Click **Select** on the vertical toolbar.
2. Select the can to contain the new the transverse member.

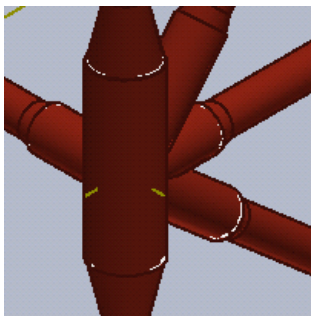


3. Select **Transverse Members** on the ribbon.
4. Select the member to add to the can.



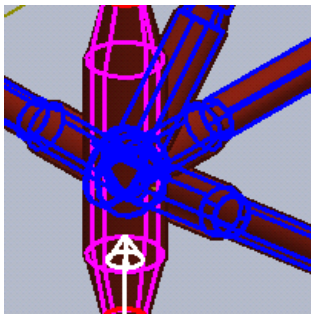
5. Click **Accept** ✓.

6. Click **Finish**.

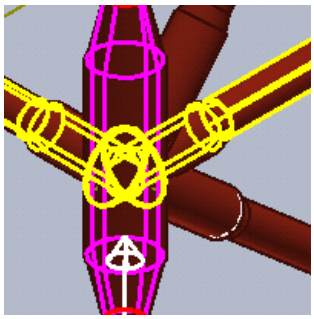


Remove a transverse member from a can

1. Click **Select** on the vertical toolbar.
2. Select the can containing the transverse member you want to remove.

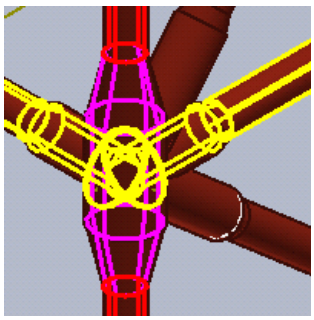


3. Select **Transverse Members** on the ribbon.
4. Select the member to remove from the can.



5. Click **Accept** ✓.

6. Click **Finish**.



Can Properties Dialog Box

Specifies the properties for the can that you are editing.

See Also

Occurrence Tab (Can Properties Dialog Box) (on page 872)
Can Rule Tab (Can Properties Dialog Box) (on page 877)
Cross Section Tab (Can Properties Dialog Box) (on page 884)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
Notes Tab (on page 817)

Occurrence Tab (Can Properties Dialog Box)

Specifies the properties for the select can.

Category

Select the properties that you want to view for the member. Currently, only Standard properties are available for BUCan designed member.

Preview

Displays a graphic indicating the different dimensions that can be specified for the selected designed member.

Standard

Name

Displays the name of the can. The can name is based on the **Name Rule** selection. If you want to type a new name for the can, in the **Name Rule** box, select **User Defined**, and then type a name for the can in the **Name** box.

Name Rule

Specify the naming rule to use to name this can. You can select one of the listed rules or select **User Defined** to specify the can name yourself in the **Name** box.

- **Default Rule** - Names the can using the syntax: BUCan<*Can Type*>-<*site ID*>-<*Index*> where: <Can Type> is the can type selected from the Catalog, <site ID> is the location ID defined in Project Management, and <Index> is a 4-digit unique index number that starts at 0001.

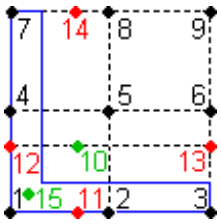
Parent System

Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.

Section Standard

Specifies the section library from which you selected the section name. Sections are defined in the reference data.

Cardinal Point

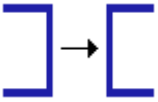


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle by which the section is rotated about the member axis.

Reflect



Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Priority

Select the priority to assign to the designed member. The priority is used to group members.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric About Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Diameter - Start

Displays the diameter of the start cone.

Diameter - End

Displays the diameter of the end cone.

Length - Start Cone

Displays the length of the start cone, which is the cone closest to the start of the supporting member.

Length - End Cone

Displays the length of the end cone.

Section Properties

Displays how section properties are defined.

Area

Displays the cross-section area for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you

need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Unit Weight

Displays the unit weight.

Length Extension

Displays the length extension, which is the minimum distance between the transverse member and the end of the tube part of the can.

Can Length

Displays the length of the can. This dimension includes the chamfer length.

Uncut Length

Displays the uncut length of the can. This dimension includes the chamfer length.

Can Type

Displays the can type that was selected from the Catalog.

L2 Hull Length

Displays the calculated L2 hull length. This dimension is measured from the outside hull of the supporting member to the start of the chamfer on the tube section.

L2 Centerline Length

Displays the calculated L2 centerline length. This dimension is measured from the center of the supporting member to the start of the chamfer on the tube section.

L3 Hull Length

Displays the calculated L3 hull length.

L3 Centerline Length

Displays the calculated L3 centerline length.

Chamfer1 Length

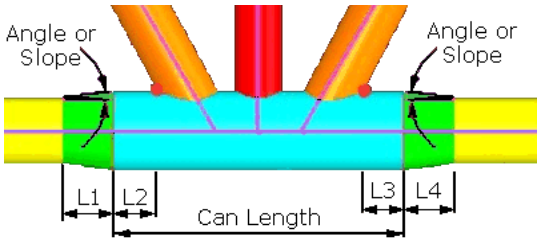
Displays the calculated Chamfer1 length.

Chamfer2 Length

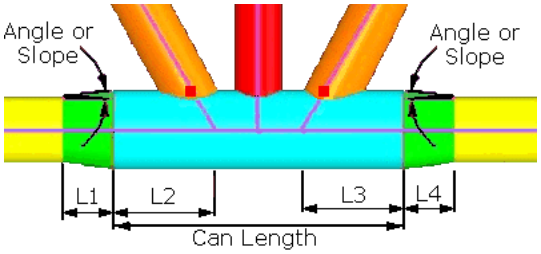
Displays the calculated Chamfer2 length.

Diameter - Tube

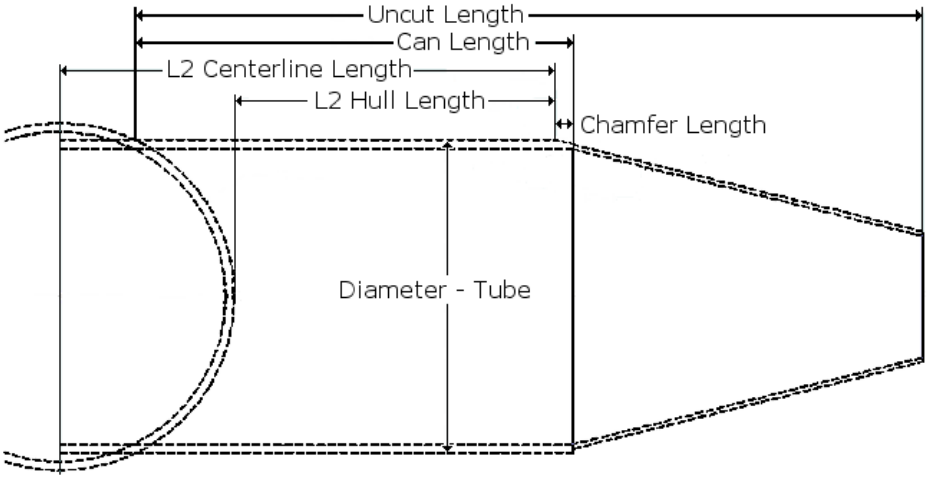
Displays the outside diameter of the tube part of the can.



L2 and L3 Lengths when using a Hull Extension Method



L2 and L3 Lengths when using a Centerline Method



Dimensions for Stub End Cans

Can Rule Tab (Can Properties Dialog Box)

Specifies the properties for the selected can rule.

Diameter Rule

Select how you want to define the diameter of the tube. You can select:

- **Outer Diameter** - Select this option to match the outside diameter of the tube with the outside diameter of the supporting leg.
- **Inner Diameter** - Select this option to match the inside diameter of the tube with the inside diameter of the supporting leg.

- **User Defined** - Select this option to specify the diameter of the tube yourself. Cones are created if you define a diameter that is significantly larger than the cross-section of the supporting leg.

Can Outside Diameter

Specifies the outside diameter of the can.

Can Inside Diameter

Specifies the inside diameter of the can.

Can Thickness

Specifies the plate thickness of the can plate.

Cone 1 Thickness

Specifies the plate thickness of cone 1.

Cone 1 Material

Select the material for cone 1.

Cone 1 Grade

Select the material grade for cone 1.

Can Material

Select the material for the can.

Can Grade

Select the material grade for the can.

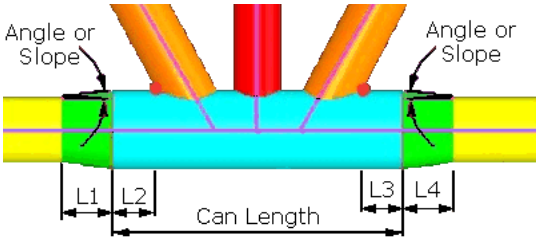
Minimum Extension Distance

Specify the minimum distance between the transverse member and the end of the can. If you type a distance that is less than the minimum extension distance that is defined in the Catalog, the software uses the distance defined in the Catalog, not the smaller distance that you specified here.

L2 Computation Method

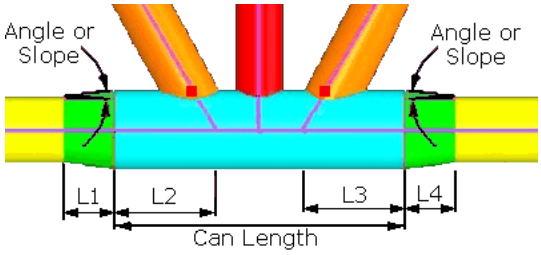
Specifies how you want Smart 3D to measure the L2 minimum distance.

- If you select **Hull-Factor**, the software calculates the outside diameter of the can times the value you type in the **L2 Factor** box. The result is used as the minimum L2 distance, which is measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.
- If you select **Hull-Length**, the software uses the distance you specify in the **L2 Length** box for L2, which is measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.



L2 and L3 Lengths when using a Hull Extension Method

- If you select **Centerline-Factor**, the software calculates the outside diameter of the can times the value you type in the **L2 Factor** box. The result is used as the minimum L2 distance, which is measured from the corresponding end of the can to the centerline of the transverse member closest to that end of the can.
- If you select **Centerline-Length**, the software uses the distance you specify in the **L2 Length** box for L2, which is measured from the corresponding end of the can to the centerline of the transverse member closest to that end of can.



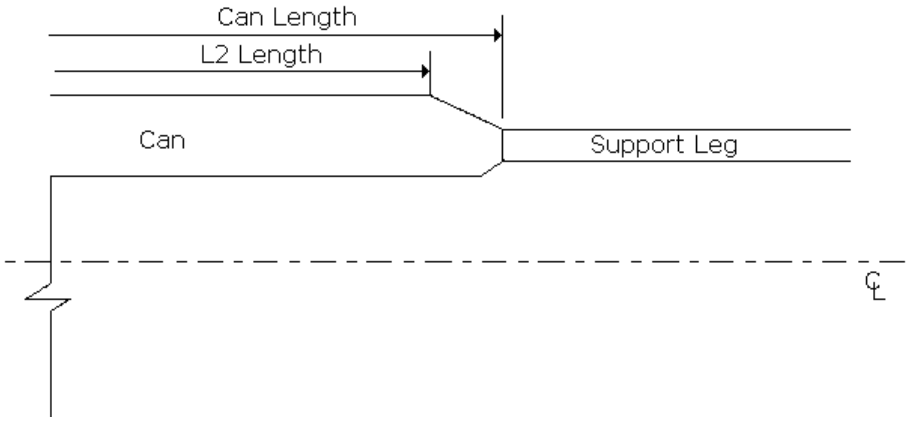
L2 and L3 Lengths when using a Centerline Method

L2 Factor

Type the factor to use to calculate L2. The default is **0.25**.

L2 Length

Type the L2 distance. This is the minimum distance that L2 can be. If there is no cone defined for that end of the can, then the L2 distance is the longer of the two chamfer lengths between the can and the support leg.

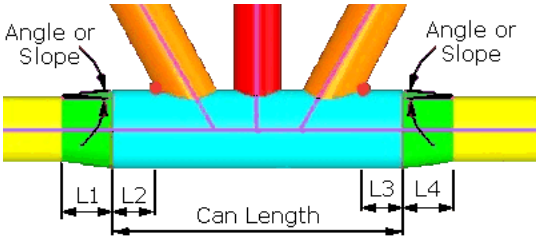


L3 Computation Method

Specifies how you want Smart 3D to measure the L3 minimum distance.

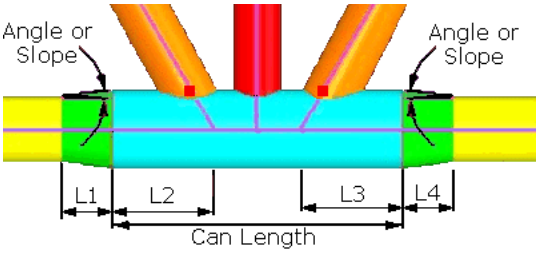
- If you select **Hull-Factor**, the software calculates the outside diameter of the can times the value you type in the **L3 Factor** box. The result is used as the minimum L2 distance which is measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.

- If you select **Hull-Length**, the software uses the distance you specify in the **L3 Length** box for L3, which is measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.



L2 and L3 Lengths when using a Hull Extension Method

- If you select **Centerline-Factor**, the software calculates the outside diameter of the can times the value you type in the **L3 Factor** box. The result is used as the minimum L3 distance, which is measured from the corresponding end of the can to the centerline of the transverse member closest to that end of the can.
- If you select **Centerline-Length**, the software uses the distance you specify in the **L3 Length** box for L3, which is measured from the corresponding end of the can to the centerline of the transverse member closest to that end of can.



L2 and L3 Lengths when using a Centerline Method

L3 Length

Type the L3 distance. This is the minimum distance that L3 can be. If there is no cone defined for that end of the can, then the L3 distance is the longer of the two chamfer lengths between the can and the support leg (see **L2 Length** above for an illustration).

L3 Factor

Type the factor to use to calculate L3. The default is **0.25**.

TIP Cone 1 is the cone nearest the start end of the supporting leg that the can is placed on.

Cone 1 Length Method

Specifies how you want to define the length of the first cone transition.

- If you select **Angle**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section, and the angle specified in the **Cone 1 Angle** box. This is shown as Angle in the figures above.
- If you select **Distance**, the length of the transition is defined explicitly in the **Cone 1 Length** box. This distance is shown as L1 in the figures above.

- If you select **Slope**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section size, and the slope specified in the **Cone 1 Slope** box. This is shown as Slope in the figures above.

Cone 1 Length

Specifies the length of the cone transition from the end of the support member to the start of the can tube. This distance is shown as L1 in the figures above. This option is available when **Cone 1 Length Method** is set to **Distance**.

Cone 1 Slope

Specifies the slope of the cone transition. The software does not create a cone if the slope you specify is greater than 100 (1/100) or less than 1^{-6}. This option is available when **Cone 1 Length Method** is set to **Slope**.

Cone 1 Angle

Specifies the angle of the cone transition. The software does not create a cone if the angle you specify is less than 1 degree or greater than 89.0 degrees. This option is available when **Cone 1 Length Method** is set to **Angle**.

TIP Cone 2 is the cone nearest the end of the supporting leg that the can is placed on.

Cone 2 Length Method

Specifies how you want to define the length of the end cone transition.

- If you select **Angle**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section, and the angle specified in the **Cone 2 Angle** box. This is shown as Angle in the figures above.
- If you select **Distance**, the length of the transition is defined explicitly in the **Cone 2 Length** box. This distance is shown as L4 in the figures above.
- If you select **Slope**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section size, and the slope specified in the **Cone 2 Slope** box. This is shown as Slope in the figures above.

Cone 2 Length

Specifies the length of the cone transition from the end of the support member to the start of the can tube. This distance is shown as L4 in the figures above. This option is available when **Cone 2 Length Method** is set to **Distance**.

Cone 2 Slope

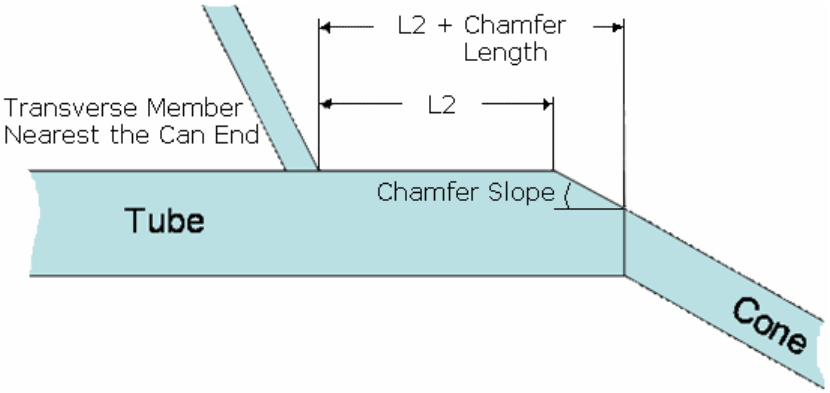
Specifies the slope of the cone transition. The software does not create a cone if the slope you specify is greater than 100 (1/100) or less than 1^{-6}. This option is available when **Cone 2 Length Method** is set to **Slope**.

Cone 2 Angle

Specifies the angle of the cone transition. The software does not create a cone if the angle you specify is less than 1 degree or greater than 89.0 degrees. This option is available when **Cone 2 Length Method** is set to **Angle**.

Chamfer Slope

Specifies the slope of the chamfer located at the end of the can where the can's tube transitions into the can's cone. The chamfer length, which is determined by the defined sloped, contributes to the length of the can part of the overall can. The value you type here is used for both ends of the can.



Roundoff Distance

Type the distance to round off the total can length. The length of the can is rounded up to the next increment of this value.

Cone 2 Thickness

Specifies the plate thickness of cone 2.

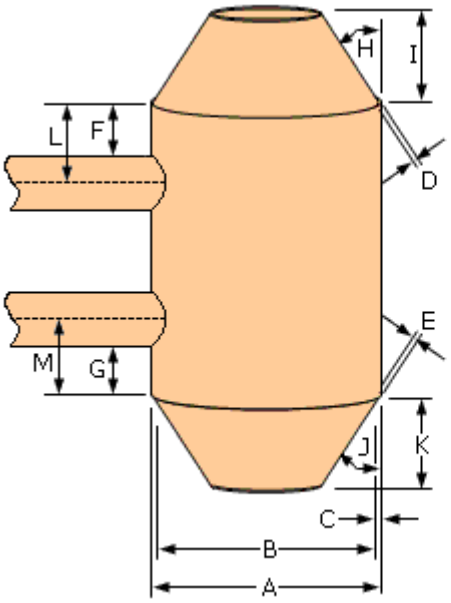
Cone 2 Material

Specifies the plate material of cone 2.

Cone 2 Grade

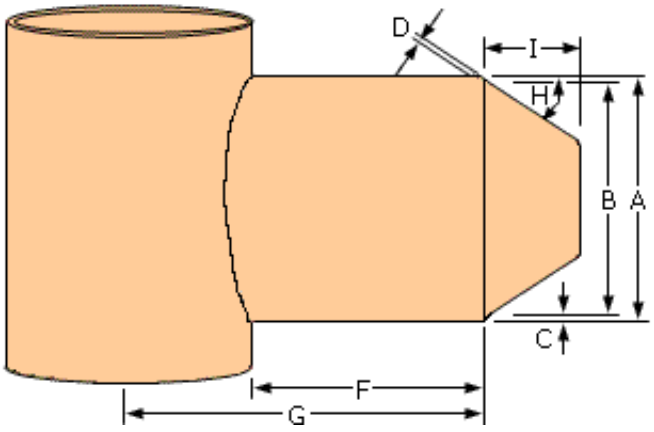
Specifies the plate material grade of cone 2.

Inline Can



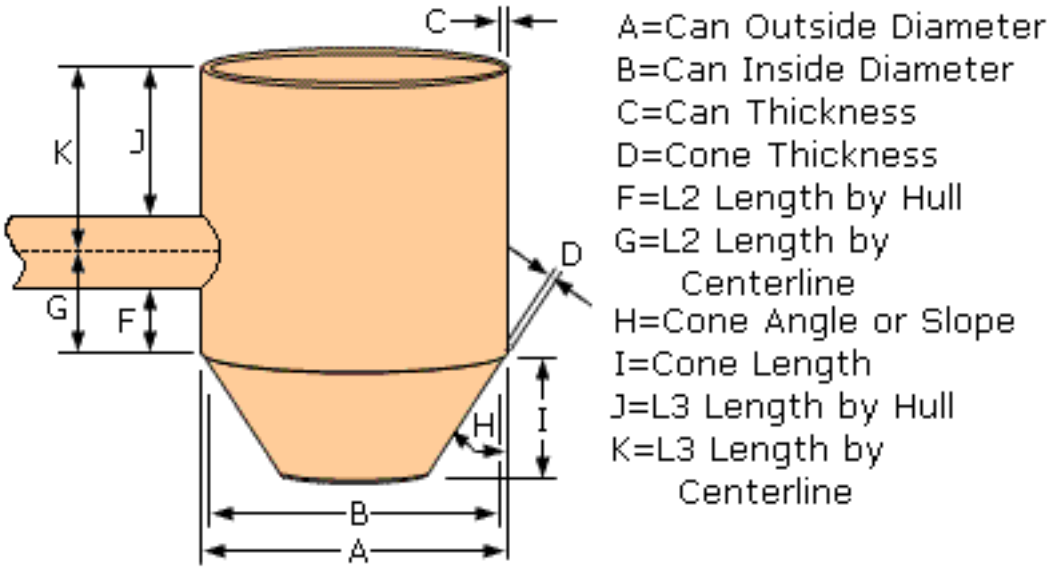
A=Can Outside Diameter
B=Can Inside Diameter
C=Can Thickness
D=Cone 1 Thickness
E=Cone 2 Thickness
F=L2 Length by Hull
G=L3 Length by Hull
H=Cone 1 Angle or Slope
I=Cone 1 Length
J=Cone 2 Angle or Slope
K=Cone 2 Length
L=L2 Length by Centerline
M=L3 Length by Centerline

Stub Can



A=Can Outside Diameter
B=Can Inside Diameter
C=Can Thickness
D=Cone Thickness
F=L2 Length by Hull
G=L2 Length by Centerline
H=Cone Angle or Slope
I=Cone Length

End Can



Cross Section Tab (Can Properties Dialog Box)

Displays the can properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one can and then selected the properties command, only the common properties between the selected cans display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

Category

Select the properties that you want to view for the member. Currently, only Standard properties are available for BUCan designed member.

Standard

Name

Displays the name of the designed member.

Symbol Definition

Displays the symbol ProgID.

Definition

Specifies the definition rule for the part class. Do not change the default value.

Parameter Rule

Specifies the parameter rule for the part class. Do not change the default value.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric About Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Diameter - Start

Displays the diameter of the start cone.

Diameter - End

Displays the diameter of the end cone.

Length - Start Cone

Displays the length of the start cone, which is the cone closest to the start of the supporting member.

Length - End Cone

Displays the length of the end cone.

Section Properties

Displays how section properties are defined.

Section Name

Type the section name. The section name must be unique across the catalog.

Short Name

Type the short name for the section.

EDI Name

Type the Electronic Data Interchange name for the section. This name is used when translating sections through CIMsteel. This property is currently not used.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Unit Weight

Displays the unit weight.

Length Extension

Displays the length extension, which is the minimum distance between the transverse member and the end of the tube part of the can.

Diameter - Tube

Displays the outside diameter of the tube part of the can.

SECTION 9

Creating Seams

The commands in the seams category provide the methods to place one or more design or planning seams in the model. By using seams, you can split plate systems or profile systems at changes in mechanical properties. When a seam is split, the logical connections between the associated plate systems or profile systems are created. A seam object splits a system into separate leaf subsystems and is always associated with one or more systems.



NOTES

- Minimum height or distances are maintained by the software for objects that use Sketch 2D, such as seams by projection. For more information, see *Minimum Distance* (on page 92).
- When you create or edit a seam by projection, the dialog box title displays the plate system name using the format **Plate System:** *<name of the plate system>*:



- You can change the default colors of seams, profile systems, reference curves, and knuckle curves to enhance visibility. For more information, see *Modify Seam Display Colors* (on page 918).



Execute Split - Automatically creates intersection seams and splits systems by intersection and design seams. The split creates new subsystems. For more information, see *Execute Split* (on page 891).



Manual Split - Places one or more intersection seams. For more information, see *Manual Split* (on page 895).



Place Split - Divides a member system into multiple member parts while maintaining the design intent of the original member system. For more information, see *Place Splits* (on page 921).



Split Seam - Splits a seam by intersecting seams. For more information, see *Split Seam* (on page 897).



Split Root System - Splits a top-level (or root) plate or profile system into multiple root systems. For more information, see *Split Root System* (on page 899).



Seam by Intersection - Creates a design or planning seam on the selected plate system by defining the landing curve of the seam as the intersection of the selected plate system and a grid plane. For more information, see *Seam by Intersection* (on page 902).



Seam by Offset - Creates a design or planning seam on the selected plate system by defining the landing curve as being offset from an existing curve (such as another seam, plate boundary, profile system, or grid plane). For more information, see *Seam by Offset* (on page 905).



Seam by Projection - Creates a design or planning seam on the selected plate system by projecting a 2D or 3D landing curve that you have sketched onto the plate system. For more information, see *Seam by Projection* (on page 908).



Seam by Table - Creates a design or planning seam on the selected plate system by defining the landing curve as coordinates in a table. For more information, see *Seam by Table* (on page 912).



Profile Seam - Creates a design or planning seam on a profile or on all profiles that belong to a plate system. For more information, see *Profile Seam* (on page 914).



Profile Knuckle - Creates a manual knuckle on the selected profile system. For more information, see *Profile Knuckle* (on page 935).

Un-Split - Merges plates into a single plate while leaving the splitting seam. Available on the Tools menu. For more information, see *Un-Split* (on page 895).

Identifying Seams in Workspace Explorer

You can identify the type of seam in the **Workspace Explorer** by the icon.

-  Design seams
-  Planning seams
-  Straking seams
-  Intersection seams

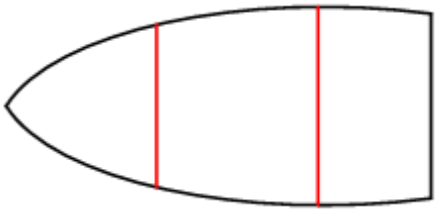
Intended Use of Seams

Intersection Seams

Created in the Molded Forms task at the intersection of two parts, such as where a deck and a bulkhead intersect. Intersection seams are automatically created by the Execute Split command in Molded Forms.

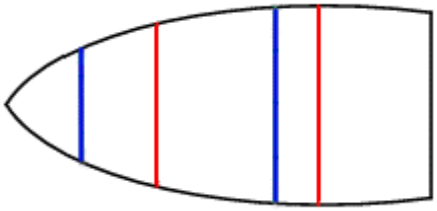
Design Seams

Created in the Molded Forms task using a seam command. A design seam is used where a design property of a system is changing, such as where the thickness of a plate changes. Leaf systems under a single parent system are created.



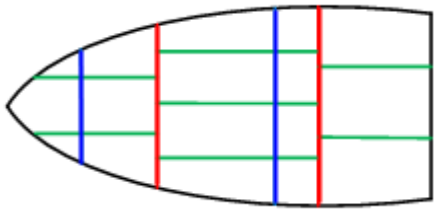
Planning Seams

Create in the Planning task using **Manage Block Intersections** or in the Molded Forms task using a seam command. A planning seam designates the limits of blocks (large segments of a ship assembled in a single location) or smaller assemblies when the limits do not coincide with design seams.



Straking Seams

Created in the Structural Detailing task to designate separate physical plates, usually due to restrictions in plate sizes that can be ordered from the mill. Plate parts under a single leaf system are created.



See Also

Knuckle Best Practices (on page 56)
Landing Curve Best Practices (on page 56)

Execute Split

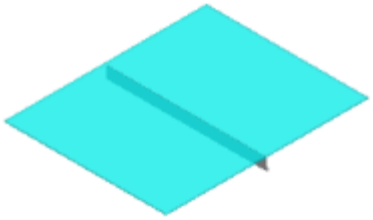
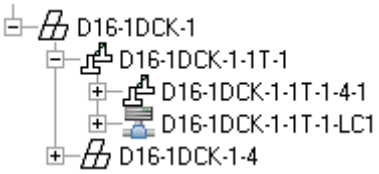
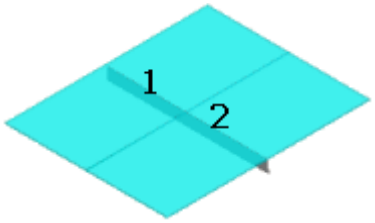
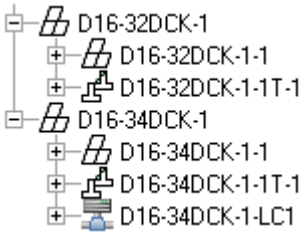
The **Execute Split** command automatically creates intersection seams and splits plate, profile, and member systems by intersection, design, and planning seams. The split creates new split subsystems and parts for plates and profiles, and new split parts for members. After you select the system that you want to split, the software displays a list of all seams and intersections for the selected system. You can then select which splits that you want performed, and, for intersections, select which systems are split and which systems are splitters.

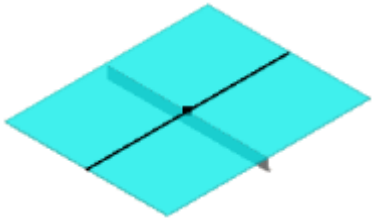
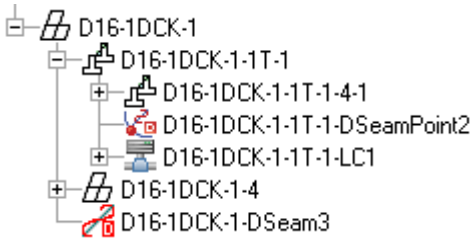
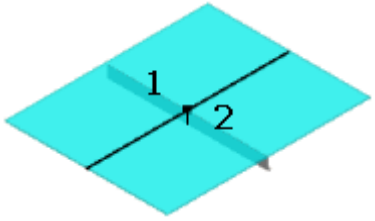
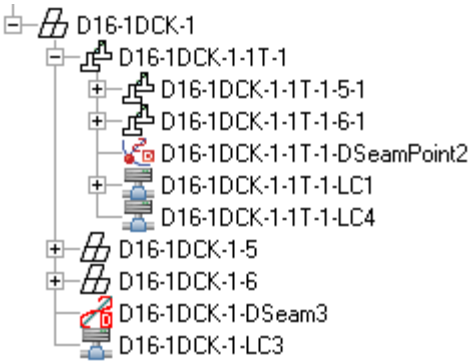
Splitting Root Systems versus Splitting with Seams

You typically use **Split Root System** to split large systems created during early design so that you can move to zone or block-based structural detail and production design.

You typically use **Execute Split** during detail design when you are refining the design with varying plate thicknesses and profile sizes, or during production design when you are creating assemblies.

The following example shows the differences between splitting systems at the root level (using **Split Root System**) and splitting systems into leaf systems using plate seams, profile seam points, and **Execute Split**.

Example	Workspace Explorer
Plate and profile systems before splitting. 	
Split Root System splits the plate and profile systems at the root level. 	

Example	Workspace Explorer
Plate and profile systems before splitting and with a plate seam (**Seam by Intersection**) and a profile seam point (**Profile Seam**). 	
Execute Split splits the plate and profile systems into leaf systems at the seams. 	

Execute Split Ribbon

The **Execute Split** ribbon displays the controls used to split systems.

Select System - Select one or more systems to split. You can select the system in the graphic view or in the **Workspace Explorer**.

Reject - Removes all the selected sets of split and splitter entities and refreshes the **Split List**.

Accept - Opens the **Split List** dialog box if it is not open, brings the split entities, if any, that you have selected in the Workspace Explorer or in the graphic view to the **Split List**, and refreshes the **Split List**. For more information, see *Split List Dialog Box* (on page 894).

Include Children - Includes children of the selected system in the splitting process.

Filter - Controls the types of objects that you can select. The following options are available:

- **All**: Allows you to select any object. This is the default value.
- **Design/Planning Seams and Knuckles**: Limits your selection in the graphic view or in the Workspace Explorer to design/planning seams and knuckles.
- **Intersection Only**: Limits your selection in the graphic view or in the Workspace Explorer to intersection objects.

***Split Best Practices* (on page 74)**

What do you want to do?

- *Split a plate system* (on page 893)
- *Split a profile or member system* (on page 893)

Split a plate system

1. Click **Execute Split** on the vertical toolbar.
2. Select the plate systems to split.

   TIPS

   - In the **Filter** box, you can select a filter to limit your selection to specific types of objects. The default filter is **All**.
   - You can select the systems either in the graphic view or in the **Workspace Explorer**.
3. Select or clear the **Include Children** option depending on your needs.
4. Click **Accept** on the ribbon.
5. Review and set the options on the **Split List** dialog box.

   TIP For more information about this dialog box, see *Split List Dialog Box* (on page 894).
6. Click **Finish** on the **Split List** dialog box.

Split a profile or member system

1. Click **Execute Split** on the vertical toolbar.
2. Select the profile or member systems to split.

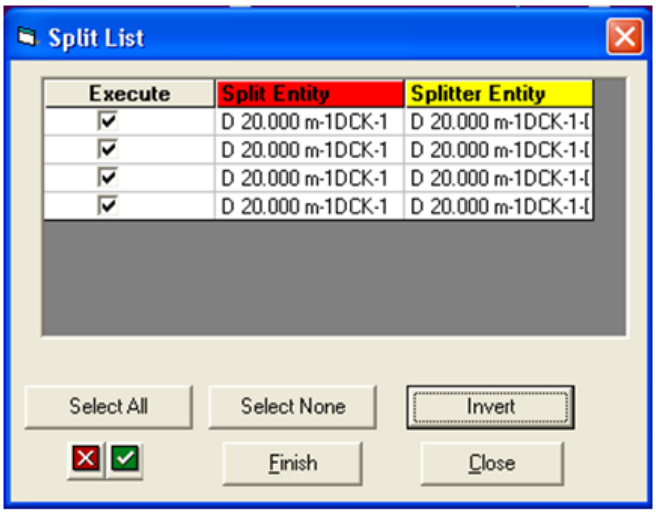
   TIPS

   - In the **Filter** box, you can select a filter to limit your selection to specific types of objects. The default filter is **All**.
   - You can select the systems either in the graphic view or in the **Workspace Explorer**.
3. Click **Accept** on the ribbon.
4. Review and set the options on the **Split List** dialog box.

   TIP For more information about this dialog box, see *Split List Dialog Box* (on page 894).
5. Click **Finish** on the **Split List** dialog box.

Split List Dialog Box

This dialog box opens during the **Execute Split** command and specifies which splits to make to the selected system.



If the system that you selected has a higher continuity number than the interfering system, then the selected system is automatically placed in the **Split Entity** column, the interfering system is placed in the **Splitter Entity** column, and the **Execute** check box is automatically selected.

If the system that you selected has a lower continuity number than the interfering system, then the selected system is automatically placed in the **Splitter Entity** column, the interfering system is placed in the **Split Entity** column, and the **Execute** check box is not selected.

If the system that you selected and the interfering system have the same continuity number, then the interference is ambiguous. The software places the selected system in the **Split Entity** column, the interfering system in the **Splitter Entity** column, and then displays the names of both systems in red. You need to review ambiguous entries and can use the **Invert** option to select the needed result.

Execute - Select which split/splitter combination to process.

Split Entity - Displays the system intersecting the higher continuity setting. If you select the **Split Entity** column, the split system highlights in red in the graphic view.

Splitter Entity - Displays the system intersecting the lower continuity setting. If you select the **Splitter Entity** column, the splitter system highlights in yellow in the graphic view.

Select All - Selects all sets of split and splitter entities.

Select None - Clears the selection of sets of split and splitter entities.

Invert - Swaps the split and splitter entities for the active row.

Reject - Removes the selected sets of split and splitter entities and refreshes the **Split List**.

Accept - Brings a split entity that you have selected in the graphic view to the **Split List** and refreshes the **Split List**.

Finish - Creates the splits using the parameters that you have defined and then updates the **Split List** to display only entities that have not been split.

Manual Split

The **Manual Split** command places one or more seams and splits on plate and profile systems created in the Molded Forms Task, and member systems created in the Structure task.

Manual Split Ribbon

Displays the controls used to place intersection seams.

System to be Split - Defines the profile or plate to be split.

Splitter System - Defines the system that will split the system to be split.

Finish - Creates intersection seams using the parameters that you have defined.

Reject - Clears the selected objects.

Accept - Accepts all selected objects to split if **System to be Split** is active. Computes the splits if **Splitter System** is active.

Split - Creates an intersecting seam. This is the default option.

Insert - Creates an inserted seam. This option is only available if you select a plate as the splitter.

Create multiple intersections seams

1. Click **Place Manual Split** on the vertical tool bar.
2. Select the plate, profile, or member system to split.
3. Click **Accept**.
4. Click **Splitter System**, and select the splitter plate or the profile system.
5. Click **Accept**.
6. Click **Split** or **Insert** to indicate the type of seam.
7. Click **Finish**.

NOTES

- If you select multiple **System to be Split**, you can select only one **Splitter System** and vice versa.
- **Insert** is only available if the splitter object is a plate.

Un-Split

The **Tools** > **Un-Split** command merges previously split plates into a single plate. The splitting seam remains in the model so that you can use the seam to split the plate again at a later time.

Un-Split Ribbon

Displays the options that you use to remove a split.

Seam Properties

Opens the **Seam Properties** dialog box, which you use to view and modify the properties of the seam that you are about to un-split in the model. For more information, see *Seam Properties Dialog Box* (on page 904).

Select

Specifies the seam that caused the split that you want to un-spit.

Un-Split

Removes the split from plates affected by the selected seam, merges the plates back into a single plate, and leaves the selected seam in the model.

Un-split a plate at a selected seam

1. Click **Tools > Un-Split** from the menu.
2. Click **Select** on the ribbon.
3. Select the seam that caused the plate split to undo.
4. Click **Un-Split**.

Split Seam

Splits a seam by intersecting seams. After you select the seam to be split, the software displays a list of all intersecting seams. You can then select which splits you want performed.

Split Seam Ribbon

Displays the options that you use to split a seam.

Select Seam(s)

Specifies the seam to be split.

Reject

Rejects the seam selection and clears the **Split List**.

Accept

Opens the **Split List** dialog box, which shows all seams intersecting the selected seam. If the dialog box is already open, refreshes the **Split List** to update the list of intersecting seams for the selected seam. For more information about the **Split List** dialog box, see *Split List Dialog Box (Split Seam)* (on page 898).

Plate Split Delay

When selected, delays the split plate update. The new position and boundaries of the design seam appear in the graphic view when this option is selected. However, the plate being split is placed on the To Do List with the message "Plate System split was stopped due to delay split" with a status of "Out of Date." You must go to the To Do List and update the plate system at a later time. This option is available only when you modify a design seam. If you do not select this option, the plate is updated as soon as you move the design seam, which can take time on a complex plate system, but no To Do List item is created for the plate.

Split a seam by intersecting seams

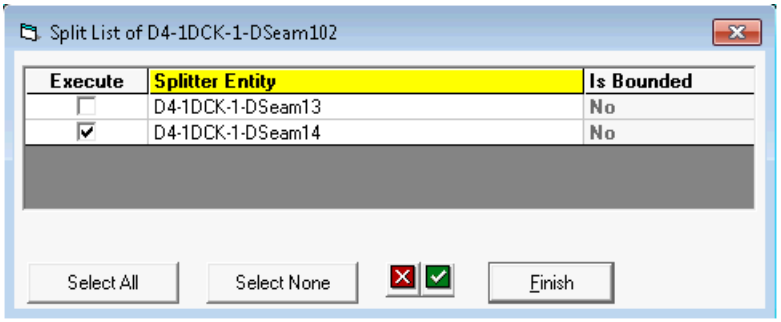
1. Click **Split Seam** on the vertical tool bar.
2. Click **Select Seam(s)**.
3. Select the seam that you want to split. The seam must be intersected by one or more seams.
4. Click **Accept**.

   *The **Split List** dialog box is displayed, showing all intersecting seams for the selected seam.*
5. In the **Execute** column, select the intersecting seams to split the seam, and click **Finish**.

   *The software immediately processes the splits. When it is completed, the **Split List** dialog box remains open and the split list is cleared.*
6. To split another seam, click **Select Seam(s)** in the ribbon to select the seam to split, and click **Accept** (either in the ribbon or in the **Split List** dialog box) to refresh the **Split List**. Select the splits to perform and click **Finish**.
7. Click **Reject** (either in the ribbon or in the **Split List** dialog box) to clear the **Split List** and clear the selection of the seam.
8. Close the **Split List** dialog box when you finish.

Split List Dialog Box (Split Seam)

This dialog box opens during the **Split Seam** command and specifies the intersecting seams used to split the selected seam.



Execute

Select which splits to process.

Splitter Entity

Displays the seams intersecting the selected seam.

Is Bounded

Specifies if the intersecting seam is bounded to the selected seam.

Select All

Selects all the splitter entities.

Select None

Clears all the splitter entities.

Reject

Rejects the seam selection and clears the **Split List**.

TIP You can directly click **Select Seam(s)** in the ribbon to select another seam to split without closing the **Split List** dialog box.

Accept

Refreshes the **Split List** to update the list of intersecting seams for the selected seam.

Finish

Splits the seam using the parameters you have defined. When the split process is complete, the **Split List** is cleared.

Split Root System

The **Split Root System** command splits a root plate system or root profile system into multiple root systems at the location of a plate system or a reference plane.

You can split a root system in the following ways:

- Select a plate system and a splitter intersecting the plate system. The plate system is split into separate root plate systems. The children of the plate system (profile systems, seams, and openings) are split into multiple objects.
- Select a profile system and a splitter intersecting the profile system. The profile system is split into separate root systems, but its parent plate system is not.

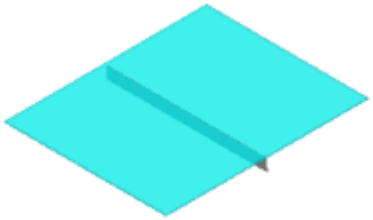
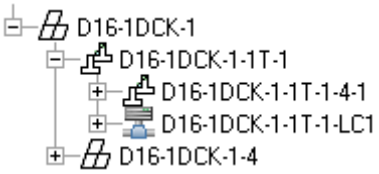
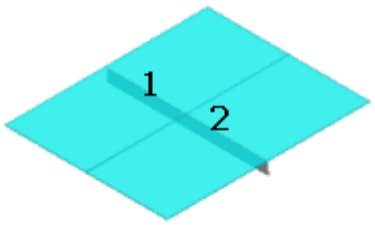
NOTE A logical connection is placed at each split plate, profile, and edge reinforcement as a child of one of the new systems.

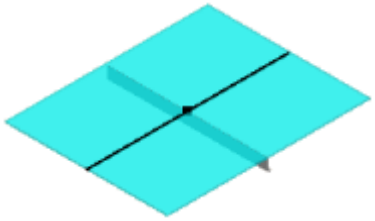
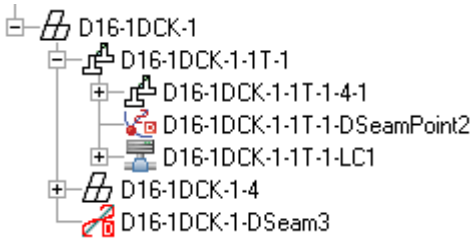
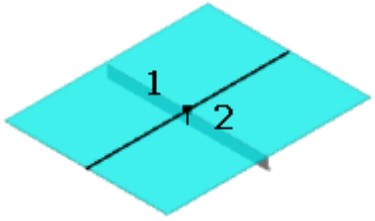
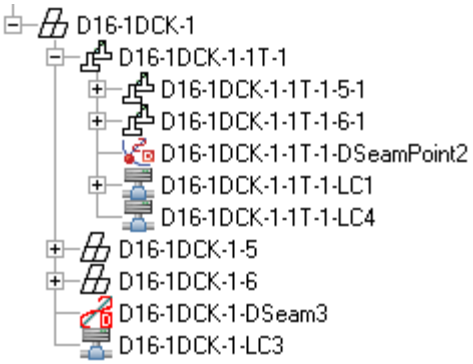
Splitting Root Systems versus Splitting with Seams

You typically use **Split Root System** to split large systems created during early design so that you can move to zone or block-based structural detail and production design.

You typically use **Execute Split** during detail design when you are refining the design with varying plate thicknesses and profile sizes, or during production design when you are creating assemblies.

The following example shows the differences between splitting systems at the root level (using **Split Root System**) and splitting systems into leaf systems using plate seams, profile seam points, and **Execute Split**.

Example	Workspace Explorer
Plate and profile systems before splitting. 	
Split Root System splits the plate and profile systems at the root level. 	

Example	Workspace Explorer
Plate and profile systems before splitting and with a plate seam (**Seam by Intersection**) and a profile seam point (**Profile Seam**). 	
Execute Split splits the plate and profile systems into leaf systems at the seams. 	

NOTE This command does not split:

- Brackets created with **Bracket Plate System**
- Child plate systems created with **Child Plate System**
- Members and built-up profiles created with **Place Linear Member**
- Tripping stiffeners created with **Tripping Stiffener System**
- Lapped plate parts created with **Lapped Plate** in the Structural Detailing task

IMPORTANT

- You have the limited ability in your current session to remove a root split and return the systems to their original state by using **Edit** > **Undo**. For more information, see *Undo* in the *Common User's Guide*.
- You cannot remove a root split or merge split systems back to their original state outside of the limits of **Edit** > **Undo** or at a later date in a different session. Because of this limitation, backing up your model before splitting is recommended.

Split Root System Ribbon

Split Root System Properties

Opens the **Split Root System Properties** dialog box, which displays log file options. For more information, see *Split Root System Properties Dialog Box* (on page 901).

Select System to Split

Specifies one or more root systems to split. Select a plate system to split the plate and its children. Select a profile system to split only the profile. Select a system (created in the Systems and Specifications task) to split all of the root plate systems within.

Select Splitter

Specifies one or more intersecting plate systems or reference planes which split the systems selected for **Select System to Split**. If you are splitting a profile, you can also select an intersecting profile system.

Finish

Splits the root systems using the defined splitters.

Reject

Clears the selections for the current step.

Accept

Accepts the selections for the current step.

Split a root system into multiple root systems

1. Click **Split Root System** on the vertical toolbar.
2. To split plate systems and their children, select one or more plate systems. Select a system (created in the Systems and Specifications task) to split all of the root plate systems within.

   OR

   To split only profile systems, select one or more profile systems.
3. Click **Select Splitter** on the ribbon.
4. Select one or more plate systems or reference planes that intersect the systems to be split. If you are splitting a profile, you can also select an intersecting profile system.
5. Click **Finish**.

   A progress bar appears showing the status of the split operation. The progress bar closes after splitting is finished.

NOTES

- Click **Cancel** on the progress bar to stop splitting of the remaining plates after the current plate is split.
- Click **Edit** > **Undo** to return all split systems to their original state.

Split Root System Properties Dialog Box

Specifies log file properties when splitting root systems. The log file can be viewed when problems occur with the split.

Log File Type

Specifies the level of detail to include in the log file. Select **Normal** or **Exhaustive**.

Display on Finish

When selected, automatically opens the log file after the system split is finished.

Log File Location

Specifies the location of the log file.

View Log

Opens the log file.

OK

Accepts changes to the values and closes the dialog box.

Cancel

Rejects changes to the values and closes the dialog box.

Seam by Intersection

The **Seam by Intersection** command creates a design or planning seam on the selected plate system by defining the landing curve of the seam as the intersection of the selected plate system and a grid plane.

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the seam boundaries. Seams must end at a plate system boundary or at another seam on the same plate system. The seam can cross an opening in the plate system.

Seam by Intersection Ribbon

Displays the options that you use to place a design or planning seam in the model.

Seam by Intersection Properties - Opens the **Seam by Intersection Properties** dialog box, which you use to view and modify the properties of the seam that you are about to place in the model. For more information, see *Seam Properties Dialog Box* (on page 904).

Plate System - Specifies the plate systems. You can select multiple plate systems to place multiple seams at the same time. The software uses the same boundaries for each seam. After placement, the seams are unrelated to each other and can be edited as individual entities.

Intersecting Objects - Specifies the objects that intersect the plate system for which you want to generate seams.

Boundaries - Defines the boundaries of the seam. You can define the boundaries by selecting objects, including other seams, in the model. If you select objects in the model and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the *Define Boundary Controls* section below.

Finish - Creates the seam using the defined parameters.

Plate Split Delay

Select this option to delay the split plate update to a more convenient time. The new position and boundaries of the design seam will appear in the graphic view when this option is selected. However, the plate being split is placed on the To Do List with the message "Plate System split was stopped due to delay split" with a status of "Out of Date." You are responsible for going to the To Do List and updating the plate system at a later time. This option is available only when

you modify a design seam. If you do not select this option, the plate is updated as soon as you move the design seam, which can take several minutes to process, but no To Do List item is created for the plate.

Seam Type

Specifies the type of seam to place. Select **Design Seam** or **Planning Seam**.

Define Boundary Controls

Available when you use the **Boundaries** option.

 **Pick Boundaries**

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

 **Solve Ambiguity**

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

 **Boundary List**

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

 **Reject** - Clears all selected plates or boundaries.

 **Accept** - Accepts all selected plates or boundaries.

Seam Type - Specifies the type of seam that you are placing. Select **Design Seam** or **Planning Seam**.

What do you want to do?

- *Place a seam by intersection* (on page 903)
- *Modify a seam* (on page 904)
- *Delete a seam* (on page 904)

Place a seam by intersection

1. Click **Seam by Intersection**  on the vertical toolbar.
2. Select the plate system to receive the seams.
3. Click **Intersecting Objects**  .
4. Select an intersecting grid plane or intersecting plate system where you want the seam.
5. Click **Boundaries**  .
6. Define the seam boundaries using one or more of the following methods. If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting seam curves.

Pick boundaries (on page 81)
Define a boundary curve (on page 82)

7. Click **Accept** .
8. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

   Solve ambiguous solution created by selected boundaries (on page 82)
9. Select whether to place a design seam or a planning seam.
10. Click **Finish** to place the seam.

Modify a seam

1. Click **Select** on the vertical toolbar.
2. Select **Seams** in the **Locate Filter**.
3. Select the seam to modify.
4. Edit the seam as needed using the ribbon bar controls.

Delete a seam

1. Click **Select** on the vertical toolbar.
2. Select **Seams** in the **Locate Filter** box.
3. Select the seam to delete.
4. Click **Edit > Delete**.

Seam Properties Dialog Box

Specifies the properties for the seam that you are placing or editing.

Main Tab (Seam Properties Dialog Box) (on page 904)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General Tab (on page 125)

Main Tab (Seam Properties Dialog Box)

Specifies the general properties of the seam.

Name

Specifies the name of the seam.

Rule

Select the naming rule to use to name the seam.

- Select **StdHierarchyChildNamingRule** to use the following syntax: <Parent Name>-<"IJSeam"><Index Number>. For example, D0-1DCK-1-IJSeam1, where D0 is the reference plane, 1DCK is an index number appended by the plate type (Deck in this

example), 1 is the Workshare Location ID, and IJSeam1 is "IJSeam" appended by an index number.

- Select **User Defined** to name the seam using the appropriate box.

Type

Specifies the type of seam that you are placing: **Design Seam** or **Planning Seam**. The list is defined by the StructSeamType codelist.

Landing Curve Definition Method

Displays how the landing curve for the seam was defined.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Seam by Offset

The **Seam by Offset** command creates a design or planning seam on the selected plate system by defining the landing curve as being offset from an existing curve (such as another seam, plate boundary, profile system, or grid plane).

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve.

You can control the boundaries of the seam by selecting a plate system, or seam that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the seam boundaries. Seams must end at a plate system boundary or at another seam on the same plate system. The seam can cross an opening in the plate system.

Seam by Offset Ribbon

Displays the options that you use to place a design or planning seam in the model.

Seam by Offset Properties - Opens the **Seam by Offset Properties** dialog box, which you use to view and modify the properties of the seam that you are about to place in the model. For more information, see *Seam Properties Dialog Box* (on page 904).

Plate System - Specifies the plate system that you want to place a seam in. You can select multiple plate systems to place multiple seams at the same time. The software uses the same information for each seam. After placement, the seams are unrelated to each other and can be edited as individual entities.

Base Curve - Specifies an existing curve or grid plane from which to offset the seam. The intersection of the grid plane and the plate system defines the base curve.

Offset - Defines the offset parameters and direction from the base curve. See the *Offset Curve Definition Controls* section below.

Boundaries - Defines the boundaries of the seam. You can define the boundaries by selecting objects, including other seams, in the model. If you select objects in the model and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the *Boundary Definition Controls* section below.

Finish - Places the seam using the defined parameters.

Reject - Clears all selected objects.

Accept - Accepts all selected objects.

Plate Split Delay

Select this option to delay the split plate update to a more convenient time. The new position and boundaries of the design seam will appear in the graphic view when this option is selected. However, the plate being split is placed on the To Do List with the message "Plate System split was stopped due to delay split" with a status of "Out of Date." You are responsible for going to the To Do List and updating the plate system at a later time. This option is available only when you modify a design seam. If you do not select this option, the plate is updated as soon as you move the design seam, which can take several minutes to process, but no To Do List item is created for the plate.

Seam Type

Specifies the type of seam to place. Select **Design Seam** or **Planning Seam**.

Offset Curve Definition Controls

Unlimited - Defines offset curves without a stop limit.

Stop Limit - Defines offset curves with a stop limit. By using a stop limit, you can select an edge, profile landing curve, connection reference curve, or seam to control the landing curves being created. If you type an offset value and the **Offset lock** is locked, then landing curves are applied at increments of the offset value until the selected stop limit is reached. If the **Offset lock** is unlocked and a value has been defined for **Count**, then the landing curves equal to the number entered in the **Count** box are equally spaced between the base curve and the stop limit.

Offset lock - Locks the **Offset** value, disabling updates of the offset value by mouse moves.

Offset - Specifies the offset distance between the base curve and the landing curve that you are defining. If you type **0**, the landing curve is place on the base curve. This box may also display the offset distance as defined by mouse movements.

Method - Specifies an offset method. You can select transverse, longitudinal, vertical, or girth.

Count - Specifies how many seams should be created, each equally spaced by increments of the offset value.

Step - Defines an increment that the system should use as you dynamically move the mouse. If you have entered a value in the **Offset** box, then this control is not used.

Boundary Definition Controls

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

What do you want to do?

- *Place a seam by offset* (on page 907)
- *Modify a seam* (on page 904)
- *Delete a seam* (on page 904)

Place a seam by offset

1. Click **Seam by Offset** on the vertical toolbar.
2. Select the plate systems to receive the seam.
3. Click **Base Curve**.
4. Select a base curve that the software is to use to compute the offset curve.
5. Click **Offset**.
6. Type an offset value in the **Offset** box.
7. Specify the offset method and count to use.
8. Select, with a mouse click in a graphic view, on which side of the base curve to apply the offset. A red arrow appears as you move the mouse indicating the direction of the offset.
9. Click **Boundaries**.
10. Define the seam boundaries using one or more of the following methods. If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting seam curves.

    Pick boundaries (on page 81)
    Define a boundary curve (on page 82)
11. Click **Accept**.
12. Select whether to place a design seam or a planning seam.

13. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)

14. Click **Finish** to place the seam.

Seam by Projection


The **Seam by Projection** command creates a design or planning seam on the selected plate system by projecting a 2D or 3D landing curve that you have sketched onto the plate system.

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the seam boundaries. Seams must end at a plate system boundary or at another seam on the same plate system. The seam can cross an opening in the plate system.

Seam By Projection Ribbon

Displays the options that you use to place a design or planning seam in the model.


Seam by Projection Properties

Opens the **Seam by Projection Properties** dialog box, which you use to view and modify the properties of the seam that you are about to place in the model. For more information, see *Seam Properties Dialog Box* (on page 904).


Plate System

Specifies the plate system for the seam. You can select multiple plate systems to place multiple seams at the same time. The software uses the same information for each seam. After placement, the seams are unrelated to each other and can be edited as individual entities.


Define using 2D Sketcher

Defines the landing curve by drawing it in the Sketch 2D environment. See the *Landing Curve Definition Controls* section below.


Landing Curve Definition by 3D

Allows creation of the seam curve directly in a graphic view by using the **Seam Palette** dialog box (similar to the **Geometric Construction Palette**), the **Geometric Construction** ribbon, and the **Geometric Construction Explorer**. This option is the 3D equivalent to Sketch 2D. For more information, see *Geometric Constructions* (on page 1027).


Boundaries

Defines the boundaries of the seam. You can define the boundaries by selecting objects, including other seams, in the model. If you select objects in the model and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the *Boundary Definition Controls* section below.

Finish

Places the seam using the defined parameters.

Landing Curve Definition Controls

Available when you use the **Define using 2D Sketcher** option.

Sketching Plane

Specify the sketching plane for the curve. This is the first step in defining the curve. See the *Define Plane Controls* section below.

Add Intersecting Item

Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.

Add Projection Item

Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the curve.

NOTE To display a grid or reference plane ruler in the Sketch 2D environment, first define an active coordinate system using **Tools** > **PinPoint**. For more information, see PinPoint in the *Common User's Guide* and *2D Grids Ruler* in the *SmartSketch Drawing Editor Help*.

Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Project Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

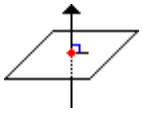
Define Plane Controls

Available when you use the **Define using 2D Sketcher** option.

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector



Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Remove/Maintain Relationships of the Plane

Maintains the relationships of the plane to the definition method and the geometry used by the definition method. For example, a plane created with **Offset from Plane** maintains the relationship of the selected plane and the **Offset** value. Otherwise, this option disables the relationships of the plane to allow moving of the plane.

NOTE When you copy a bracket and use **Edit** > **Paste** > **Delete Optional** to remove a boundary, **Remove/Maintain Relationships of the Plane** is not visible. You must first click **Select Bracket Supports** to allow the command to determine the applicable options.

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Boundary Definition Controls

Available when you use the **Boundaries** option.

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Plate Split Delay

Select this option to delay the split plate update to a more convenient time. The new position and boundaries of the design seam will appear in the graphic view when this option is selected. However, the plate being split is placed on the To Do List with the message "Plate System split was stopped due to delay split" with a status of "Out of Date." You are responsible for going to the To Do List and updating the plate system at a later time. This option is available only when you modify a design seam. If you do not select this option, the plate is updated as soon as you move the design seam, which can take several minutes to process, but no To Do List item is created for the plate.

Seam Type

Specifies the type of seam to place. Select **Design Seam** or **Planning Seam**.

What do you want to do?

- *Place a seam by projection* (on page 912)
- *Define landing curve by 2D* (on page 87)
- *Define landing curve by 3D* (on page 88)
- *Modify a seam* (on page 904)
- *Delete a seam* (on page 904)

Place a seam by Projection

1. Click **Seam by Projection** on the vertical toolbar.
2. Select the plate system for the seam.
3. Define the landing curve using one of the following methods:

   Define landing curve by 2D .

   Define landing curve by 3D .
4. Click **Boundaries** .
5. Define the seam boundaries using one or more of the following methods. If you do not select boundaries, the software applies the entire length of the base curve to the resulting seam curves.

   Pick boundaries (on page 81)
   Define a boundary curve (on page 82)
6. Click **Accept** .
7. Select if you are placing a design seam or a planning seam.
8. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

   Solve ambiguous solution created by selected boundaries (on page 82)
9. Click **Finish** to place the seam.

Seam by Table

The **Seam by Table** command creates a design or planning seam on the selected plate system by defining the landing curve as coordinates in a table.

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the seam boundaries. Seams must end at a plate system boundary or at another seam on the same plate system. The seam can cross an opening in the plate system.

Seam By Table Ribbon

Displays the options that you use to place a design or planning seam in the model.

Seam by Table Properties - Opens the **Seam by Table Properties** dialog box, which you use to view and modify the properties of the seam that you are about to place in the model. The initial settings default from the parent plate system. For more information, see *Seam Properties Dialog Box* (on page 904).

Plate System - Specifies the plate system for the seam. You can select multiple plate systems to place multiple seams at the same time. The software uses the same information for each seam. After placement, the seams are unrelated to each other and can be edited as individual entities.

Landing Curve Table - Opens a table where you can define the landing curve coordinates for the seam. For more information, see *Landing Curve Table Dialog Box* (on page 760).

Boundaries - Defines the boundaries of the seam. You can define the boundaries by selecting objects, including other seams, in the model. If you select objects in the model and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the *Boundary Definition Controls* section below.

Finish - Places the seam using the defined parameters.

Plate Split Delay

Select this option to delay the split plate update to a more convenient time. The new position and boundaries of the design seam will appear in the graphic view when this option is selected. However, the plate being split is placed on the To Do List with the message "Plate System split was stopped due to delay split" with a status of "Out of Date." You are responsible for going to the To Do List and updating the plate system at a later time. This option is available only when you modify a design seam. If you do not select this option, the plate is updated as soon as you move the design seam, which can take several minutes to process, but no To Do List item is created for the plate.

Seam Type

Specifies the type of seam to place. Select **Design Seam** or **Planning Seam**.

Boundary Definition Controls

Available when you use the **Boundaries** option.

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

Reject - Clears all selected objects.

Accept - Accepts all selected objects.

Landing Curve Table Dialog Box (on page 760)

What do you want to do?

- *Place a seam by table* (on page 914)
- *Modify a seam* (on page 904)
- *Delete a seam* (on page 904)

Place a seam by table

1. Click **Seam by Table** on the vertical toolbar.
2. Select the plate system for the seam.
3. Click **Landing Curve Table**.
4. Define the landing curve coordinates in the table. For more information about defining coordinates, see *Landing Curve Table Dialog Box* (on page 760).
5. Click **Close**.
6. Click **Boundaries**.
7. Define the seam boundaries using one or more of the following methods. If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting seam curves.

   Pick boundaries (on page 81)
   Define a boundary curve (on page 82)
8. Click **Accept**.
9. Select whether to place a design seam or a planning seam.
10. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
11. Click **Finish** to place the seam.

Profile Seam

The **Profile Seam** command creates a design seam or a planning seam on a profile or on all profiles that belong to a plate system. You can select individual profile systems to split, or you can select a plate system and thereby implicitly select all the profile systems that belong to the selected plate system to split. Optionally, you can automatically split the profile systems after the profile seams are placed.

Profile Seam Ribbon

Displays the options that you use to split a profile.

Properties

Opens the **Profile Seam Properties** dialog box, which you can use to define properties for the seam that you are about to place. For more information, see *Profile Seam Properties Dialog Box* (on page 917).

Profile System

Specifies the profile system or the plate system that has associated profile systems to split. You can select stiffener profile systems or linear beam profile systems.

The following option displays for this step:

Select - Specifies the profile selection method. Select **Plate Systems** to select profiles by identifying their parent plate system. Select **Profile Systems** to identify profiles directly.

References

Specifies objects that intersect the profile system at which to place the split. You can select objects in either the graphic view or in the **Workspace Explorer**. Objects that you can select as references include: reference curves, grid planes, plate systems, edge reinforcement systems, seams (design, planning, and straking), profile systems (landing curve), and edge ports including openings.

The following options display for this step:

Method - Specifies the seam placement method.

Offset - Places the seam at a distance from the selected reference objects, using the selected offset method.

Along Profile - Places the seam along the profile landing curve. This method displays the **PinPoint** ribbon. For more information, see PinPoint.

Offset Method - Specifies the offset measurement method. This option is only available when you select **Offset** in the **Method** box.

Girth - Measure the offset value along the surface of the profile's parent plate system.

Vertical - Measure the offset value in the direction of the z-axis of the global coordinate system.

Transverse - Measure the offset value in the direction of the y-axis of the global coordinate system.

Longitudinal - Measure the offset value in the direction of the x-axis of the global coordinate system.

Offset Value - Defines the offset value. You can specify the direction of the offset by typing a plus (+) or minus (-) sign in front of the offset value. A positive value is offset in the positive direction of the closest main coordinate system axis. If you type **0**, the seam is placed at the reference. This option is only available when you select **Offset** in the **Method** box.

Finish

Splits the profile using the parameters that you have defined.

Reject

Rejects the selected objects.

Accept

Accepts the selected objects and advances the command to the next step.

Seam Type

Specifies the profile seam type. Select **Design Seam** or **Planning Seam**.

Auto Split

When selected, splits the profile system after placing the profile seam.

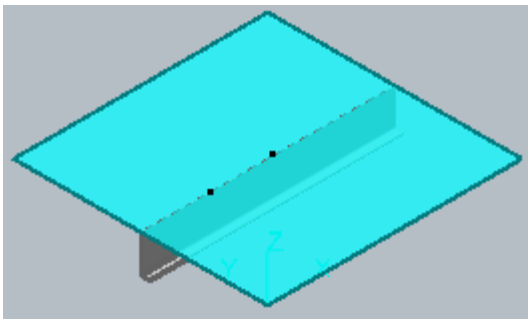
What do you want to do?

- *Place a profile seam by offsetting from a reference* (on page 916)
- *Place a profile seam at a distance along the profile* (on page 916)

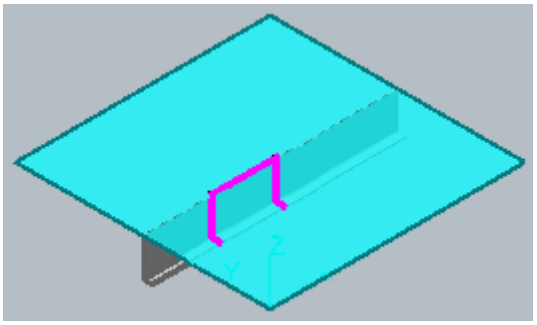
Place a profile seam by offsetting from a reference

1. Click **Profile Seam** on the vertical toolbar.
2. Select **Design Seam** or **Planning Seam** in the **Select** box.
3. To split individual profiles, select **Profile Systems** in the **Select** box, and then select one or more profile systems.
4. To split all profiles on a plate system, select **Plate Systems** in the **Select** box, and then select one or more plate systems.
5. Select one or more model objects or reference planes to use as reference for the offset.
6. In the **Method** box, select **Offset**.
7. To define the profile seam location, select the offset method and type an offset value.
8. Click **Finish**.

   The software places the profile seams.



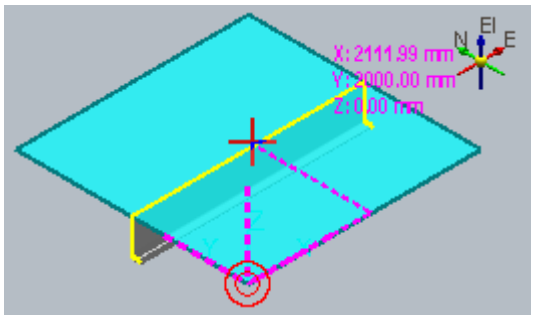
NOTE *If you select **Auto Split** on the ribbon, the software also splits the profile when you click **Finish**.*



Place a profile seam at a distance along the profile

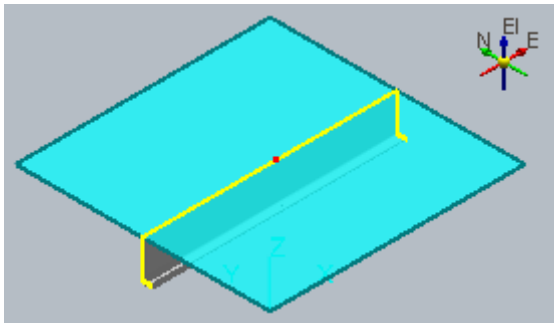
1. Click **Profile Seam** on the vertical toolbar.
2. Select **Design Seam** or **Planning Seam** in the **Select** box.
3. Select **Profile Systems** in the **Select** box, and then select a profile system.
4. Click **References**.
5. In the **Method** box, select **Along Profile**.

*The **PinPoint** ribbon displays, and the dotted magenta line of **PinPoint** displays on the profile landing curve.*

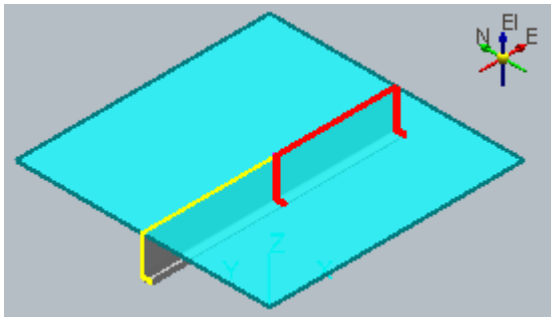


6. To define the profile seam location, click a location in the graphic view, or type X, Y, and Z values. For more information on using **PinPoint**, see Place objects using rectangular coordinates.
7. Click **Finish**.

   The software places the profile seam.



NOTE *If you select **Auto Split** on the ribbon, the software also splits the profile when you click **Finish**.*



Profile Seam Properties Dialog Box

Specifies the properties for the profile seam.

Main Tab (Profile Seam Properties Dialog Box) (on page 917)

Main Tab (Profile Seam Properties Dialog Box)

Specifies the general properties for the profile seam.

Name - Displays the name that is assigned to the profile seam.

Rule - Specifies how the name is created.

- Select **StdHierarchyChildNamingRule** to use the following syntax: <Parent Name>-<Index Number><Profile Type>-<Index Number>-<"IJSeamPoint"><Index Number>. For example, D0-1DCK-1-3T-1-IJSeamPoint1, where D0-1DCK-1 is the parent name, 3T is the profile index number and type, 1 is an index number, and IJSeamPoint1 is "IJSeamPoint" appended by an index number.
- Select **User Defined** if you want to name the seam yourself by using the appropriate box.

Type - Displays the current object type. You can select the Design Seam or Planning Seam.

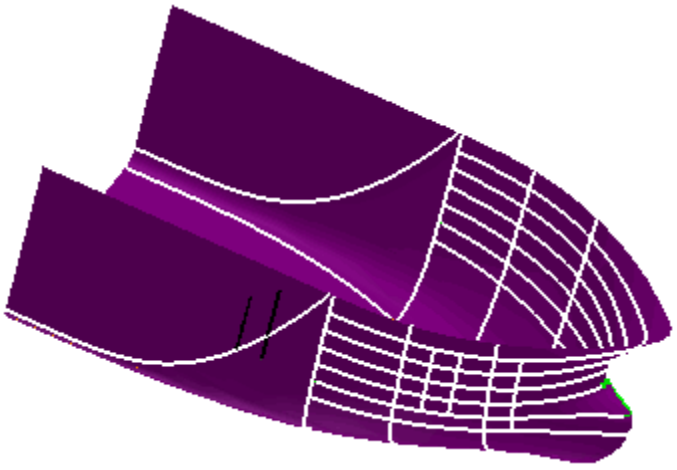
Parent System - Displays the name of the parent system of the profile seam.

Modify Seam Display Colors

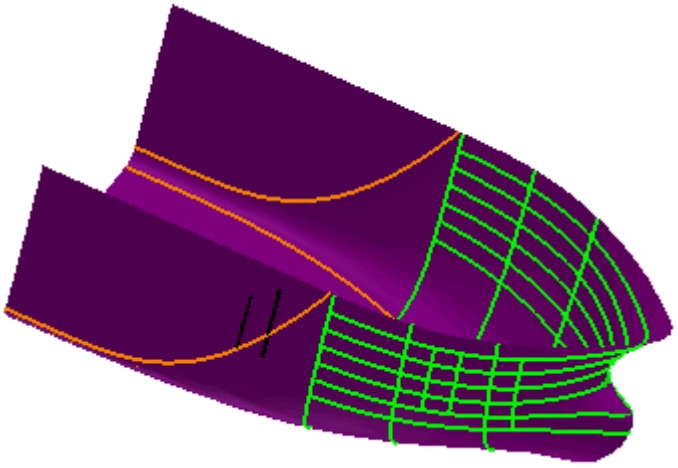
Use this procedure to create style rules to customize the display colors of landing curves. You can use the procedure to set different colors for the landing curves of:

- Seams
- Profile Systems
- Reference Curves
- Plate Knuckle Curves

For example, before creating style rules, reference curves and design seams are white:



After creating separate style rules for reference curves and design seams, they appear in different colors:



Use the following steps to customize curve display colors.

Select the View Aspect

1. Click **Format** >**View**.

   *The **Format View** dialog box opens.*

2. In the **Selected Aspects** box, click **Reference Geometry**.

   NOTE By default, **Simple physical** is also selected.

3. Click **OK**.

Create a Style Rule

1. Click **Format** > **Surface Style Rules**.

   *The **Surface Style Rules** dialog box opens.*

2. Click **New**.

   *The **Surface Style Rule Properties** dialog box opens*

3. Type a name for the new surface style rule in the **Rule name** box.
4. In the **Filter** list, click **More** to create a new filter.

   *The **Select Filter** dialog box opens.*

5. Click **New Filter**.

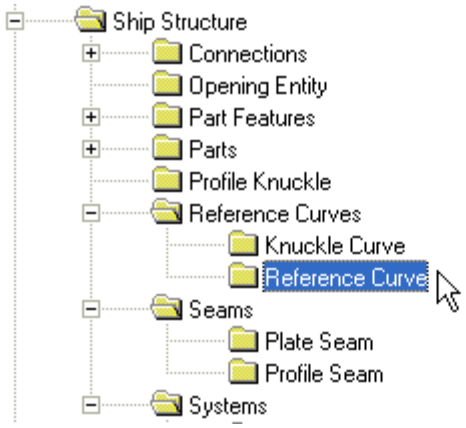
   *The **New Filter Properties** dialog box opens.*

   NOTE If you are an administrator, you can first select **Ship Filters**. The new filter is then available to all users.

6. On the **Object Type** tab, expand the hierarchy **Ship Structure**, and select one of the following:

   - Ship Structure/ Reference Curves/**Reference Curve**

- Ship Structure/ Reference Curves/**Knuckle Curve**
- Ship Structure/Seams/**Plate Seam**
- Ship Structure/Systems/Profile Systems/**Stiffener System**



7. Click **OK**.
8. Click **OK** in the **Select Filter** dialog box.
9. In the **Surface Style Rule Properties** dialog box, select a color in the **Style applied** box.
10. Click **OK**.

    *The new rule appears in the **Style rule library** list.*

Add the Rule to the Workspace

1. Select the rule in the **Style rule library** list.
2. Click **Add**.
3. Click **OK**.

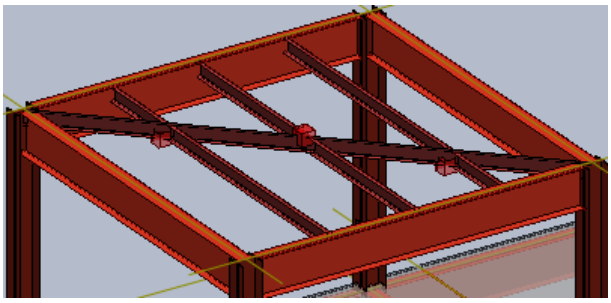
See Also

In the Common User's Guide:
Format View
Format a View
Surface Style Rules
Create a Surface Style Rule
Create a New Filter
Add a Surface Style Rule to the Workspace

SECTION 10

Place Splits

Divides a member system (created in the Structure task) into multiple member parts. The resulting member system is a set of continuous member parts that move as a single entity. This splitting is useful when you want to resolve the interference between two intersecting structural objects, such as pair of cross braces, or when you want to split the columns in your model at certain elevations. You can split member systems at another member system, at a point along the member system, a grid plane, an elevation plane, deck, bulkhead, or plate system.



Only member systems can be split using the command. However, the object used to split the member system can be another member system, a grid plane, an elevation plane, deck, bulkhead, or plate system.

To reconnect a split member system, delete the split connections. The split connects are child objects of the member system that they are splitting.

If after you split a member system into multiple member parts you want to delete an individual member part, you will need to create a member system parent for that member part using the **Convert** option. Set the locate filter to **Parts** and select the member part, then click **Convert** on the modify ribbon. For more information, see *Modify Linear Member Part Ribbon* in *Place Linear Member Systems* (on page 787).

Member End Releases

If a column is split, the upper end of the lower column and the lower end of upper column has all end releases fixed. The lower end of the lower column and the upper end of the upper column do not change.

For beams and braces that are split, end releases on either side of the split are pinned for rotation (RY and RZ). The free end of a cantilevered beam or brace has fully fixed member end releases.

Boundary Conditions

Boundary Conditions remain on the ends for which they were defined. For example, if end 1 of the original member was fully supported (X, Y, Z, RX, RY, RZ) and end 2 was supported only in RX and RZ, then end 1 of new member 1 gets fully supported and end 2 of new member 2 gets the RX and RZ support. The new interior ends (end 2 for member 1 and end 1 for member 2) do not have any supports defined.

For more information about boundary conditions, refer to the Structural Analysis task documentation.

Loads

Concentrated loads stay where they were placed (the physical location along the original member part length). The absolute or relative placement value is recalculated based on the new member part length.

Distributed loads are split into two distributed loads with the same magnitude (one distributed load for each new member part).

Partially distributed loads do one of two things based on the split location. If the split location is between the partially distributed load's end points, the load is split into two partially distributed loads - one on each new member part on either side of the split location.

If the split location is outside of the partially distributed load's end points, the load is unaffected other than having the end points' absolute or relative placement values recalculated based on the new member part length.

For more information about loads, refer to the Structural Analysis task documentation.

Fireproofing

When a member with fireproofing is split, Smart 3D creates new fireproofing on the two new split members with the same setback references as the original member's fireproofing. In addition, the software adjusts the setback distances so that the fireproofing maintains the same geometric start and end position and looks solid throughout.

If you delete a split, the fireproofing is merged using the properties from the existing fireproofing. You must edit the resulting fireproofing and adjust the setbacks and other properties as needed.

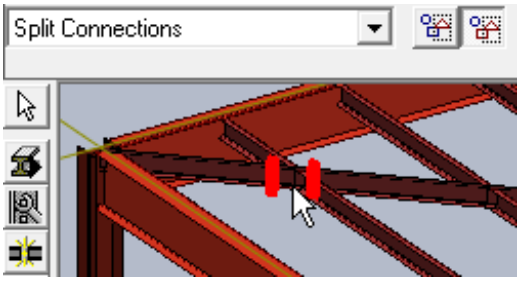
Designed Members

When splitting designed members, you need to be aware of these limitations:

- You can split designed members only by a rolled member, a specified point, a grid plane, or another designed member.
- If you split the designed member by a rolled member, then each plate in the designed member has a logical connection to the rolled member.
- If you split the designed member by a grid plane, then the plate parts on one side of the grid plane have a logical connection to the designed member part on the other side of the grid plane.
- The split always places a square end-cut on the designed member. This square cut is replaced by the appropriate end-cut when the designed member is detailed.
- Hangers, sketched boundaries, openings, features, chamfers, detailed plate parts, intersection seams connected to the designed member's plate systems *before* the split are deleted when the split is complete. Child plate systems and other plate systems referencing a designed member's plate system are put on the To Do List. You must redefine the reference relationships to remove them from the To Do List.
- Plate systems bounded by the designed member's plate systems before the split become unbounded when the split is complete. Stiffeners and edge reinforcements on these plate systems are also deleted upon completion of the split.

Locating Split Connections

Split connections do not display in the model. However, if you set the **Locate Filter** to **Connections**, you can locate and select split connections for review and editing. Split connections are located at the ends of member parts where the split is located and appear as circles when you move the cursor over them.



When you select a split connection, the software displays the parent member system of the split connection in the ribbon. Select the **Edit > Properties** command to edit the split connection properties. You cannot measure distances using the split connection as an end point when using the **Tools > Measure** command.

Place Split Ribbon

Specifies the properties for the split that you are placing or editing.

Split Properties

Activates the **Split Properties** dialog box. For more information, see *Split Connection Properties Dialog Box* (on page 929).

Place

Select how to define the split location.

- Select **By Object** to split the member system by another model object that you identify.
- Select **By Point** to split the member system at a point along the member system that you identify.

Finish

Splits the member systems using the options that you have specified.

Cancel

Clears the selected objects.

Accept

Accepts the selected objects and moves you to the next step in the command.

By Object Options

Split Members

Select the members to split. Remember, if you set **Split Status** to **By Rule** then you cannot split members that have **Continuous** set for the **Continuity Type**. You can split only members set to **Intercostal**.

Splitting Members

Select the objects at which to split the member systems. The objects can be another member system, a grid plane, an elevation plane, deck, or a bulkhead.

Split Status

Specifies the method to determine which member system splits another member system.

- **By Rule** - The software looks at the **Continuity Type** and **Continuity Priority Number** values defined on the *Member System Tab (Member System Prismatic Properties Dialog Box)* (on page 813) to determine which member system to split. Member systems set to **Continuous** split the other member system. Member systems set to **Intercostal** are split by the other member system. The **Continuity Priority Number** is used to select which member system is split when two member systems intersect, but both have **Intercostal** for the **Continuity Type**. Member systems with a lower continuity priority (1, 2, 3, for example) split member systems with a higher continuity priority (7, 8, 9, for example).
- **Split First** - The member systems that you identified during the **Split Members** step are split by the member systems that you identified during the **Splitting Members** step.
- **Split Second** - The member systems that you identified during the **Splitting Members** step are split by the member systems that you identified during the **Split Members** step.
- **Split Both** - All member systems that you have identified in both steps are split against each other.
- **Split None** - No member systems are split, but the interference between the two objects is suppressed.

By Point Options

Split Member

Select the member to split.

Splitting Point

Defines the split location point.

Location Definition

Specifies how the split location is calculated. The location can be defined as **Ratio** or **Distance**.

- A ratio location can be thought of as a distance percentage along the member. For example, a split placed at a relative distance of 0.333 is located at one-third the member length measured from the member start. A split placed at a relative distance of 0.5 is located at the middle of the member.
- A distance value is the actual distance from the start or end of the member. The software verifies that the absolute distance that you specify is not past the end of the member.

Offset Reference

Select whether to measure the split location from the start or the end of the member.

Offset Value

Specify the position of the split as either the ratio distance along the member or the absolute

distance (as defined by **Location Definition**.)

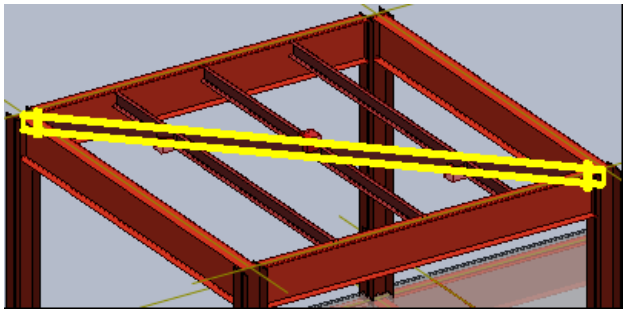
What do you want to do?

- *Split a member that intersects another member* (on page 925)
- *Split a cross brace* (on page 926)
- *Split columns at a plane* (on page 927)
- *Split a member at a point along member* (on page 928)
- *Set member split priority* (on page 928)
- *Remove a split* (on page 929)

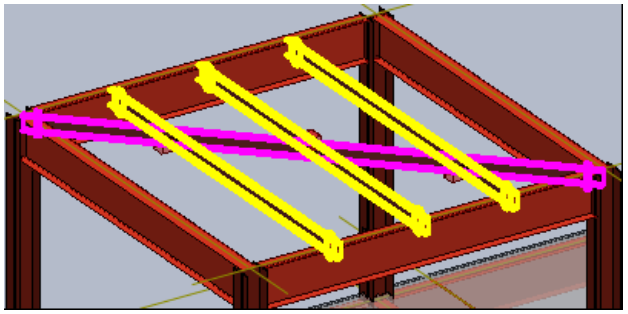
Split a member that intersects another member

1. Click **Place Split** on the vertical toolbar.
2. In the **Place** box, select **By Object**.
3. In the **Split Status** box, select **Split First**.

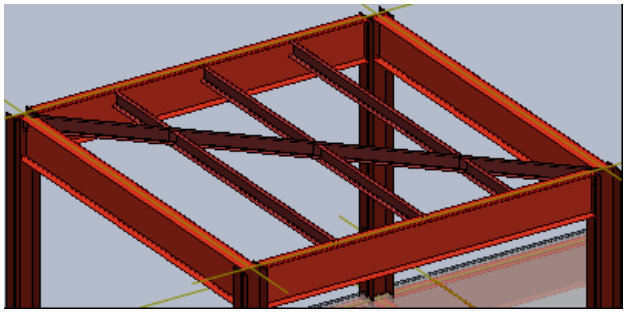
   TIP If the **Continuity Type** and **Continuity Priority Number** values have been set for the member systems, you can use the **By Rule** option to have the software automatically determine which member system to split. For more information, see *Place Split Ribbon* in *Place Splits* (on page 921).
4. Select the member to split.



5. Click **Accept** ✓.
6. Select the splitting member.

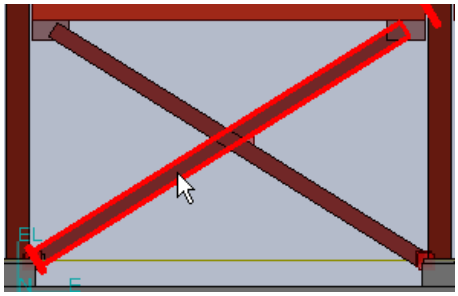


7. Click **Finish**.

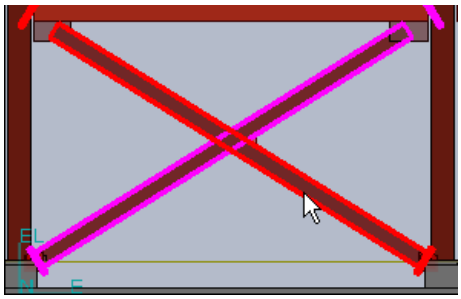


Split a cross brace

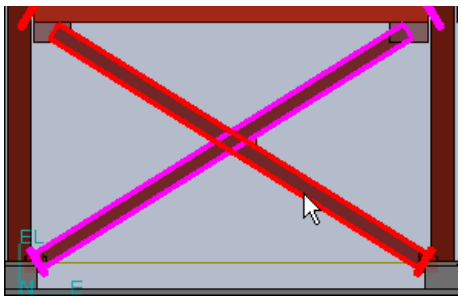
1. Click **Place Split** on the vertical toolbar.
2. In the **Place** box, select **By Object**.
3. In the **Split Status** box, select **Split First**.
4. Select the cross brace to split.



5. Click **Accept** ✓.
6. Select the other cross brace.

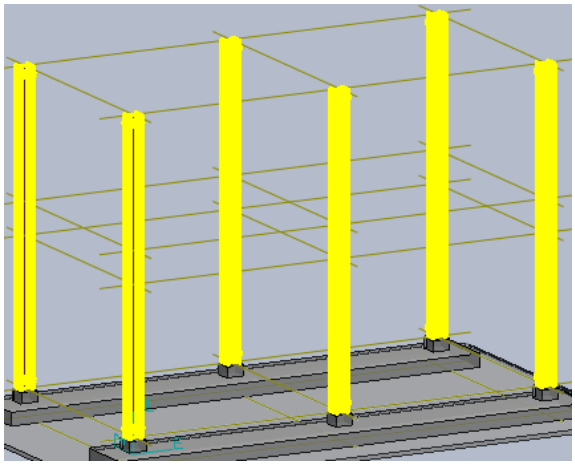


7. Click **Finish**.



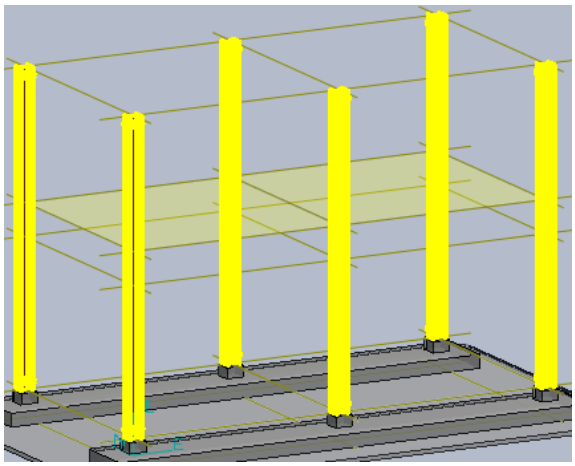
Split columns at a plane

1. Click **Place Split** on the vertical toolbar.
2. In the **Split Status** box, select **By Rule**.
3. Select the columns to split.



4. In the **Workspace Explorer**, select the elevation plane at which to split the columns.

TIP You can also select the elevation plane by selecting it on the Z-axis ruler. Select **View > Rulers** to turn on rulers.



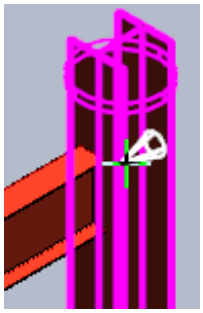
5. Click **Accept** ✓.
6. Click **Finish**.

NOTE Use the Grids task to create elevation planes.

Split a member at a point along member

1. Click **Place Split** on the vertical toolbar.
2. In the **Place** box, select **By Point**.
3. Select the member to split.

   An arrow indicates the split point.



4. In the **Location Definition** box, select **Distance** or **Ratio**.
5. Define the location for the split using the **Offset Reference** and **Offset Value** boxes.
6. Click **Finish**.

Set member split priority

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Members System**.

3. Select the member system to edit.
4. Click **Properties** .
5. Set the **Continuity Type** and **Continuity Priority Number** options as required. See *Member System Tab (Member System Prismatic Properties Dialog Box)* (on page 813) for information on these two properties.
6. Click **OK**.

Remove a split

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Split Connections**.
3. In a graphic view, select the split to remove.
4. Click **Delete** .

Split Connection Properties Dialog Box

Specifies the properties for the split connection that you are editing.

See Also

General Tab (Split Connection Properties Dialog Box) (on page 929)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
Notes Tab (on page 817)

General Tab (Split Connection Properties Dialog Box)

Specifies the properties for the split connection.

Category

Select the type of properties to view for the selected split connection.

Name

Specifies the name of the split connection.

Name Rule

Select the name rule to use to name the split connection.

- **Default Name Rule** - Names the split connection using the format: <split status>-<location>-<index> where <split status> is the Split Status that was used to create the split connection, <location> is the global workshare location, and <index> is a unique index number that starts at 0001. For example, Split First-1-0004.
- **Unique Name Rule** - Names the split connection using the format: <split status>-<location>-<index> where <split status> is the Split Status that was used to create the split connection, <location> is the global workshare location, and <index> is a unique index number that starts at 0001. For example, Split First-1-0004.
- **User Defined** - Select this name rule to name the split connection yourself in the **Name** box.

Parent

Displays the parent member system to which the split belongs.

Split Parent Status

Specifies the method Smart 3D uses to determine the member system that splits another member system. The list is defined by the SplitStatus codelist.

By Rule indicates that the software looks at the **Continuity Type** and **Continuity Priority Number** values defined on the *Member System Tab (Member System Prismatic Properties Dialog Box)* (on page 813) to determine the member system to split. Member systems set to **Continuous** split the other member system. Member systems set to **Intercostal** are split by the other member system. The **Continuity Priority Number** specifies the member system that is split when two member systems intersect, but both have **Intercostal** for the **Continuity Type**. Member systems with a lower continuity priority (1, 2, 3, for example) split member systems with a higher continuity priority (7, 8, 9, for example).

Split First indicates that the member systems that you identified during the **Split Members** step are split by the member systems that you identified during the **Splitting Members** step.

Split Second indicates that the member systems that you identified during the **Splitting Members** step are split by the member systems that you identified during the **Split Members** step.

Split Both indicates that all member systems that you have identified in both steps are split against each other.

Split None indicates that no member systems are split, but the interference between the two objects is suppressed.

Offset

Specify the offset from the intersecting plane or surface to place the split. You can specify a positive or negative number. This option is only available after the split connection is placed when defining a split using a grid plane, elevation plane, or surface as the splitting object.

See Also

Split Connection Properties Dialog Box (on page 929)

SECTION 11

Creating Knuckles

Knuckles are created automatically as children of extruded plate systems and of profile systems by projection when there is an acute discontinuity (an angle with no radius) on the extrusion or landing curve. These knuckles can be modified through the dialog boxes shown below. Profile knuckles can also be added manually.

NOTE You can change the default colors of knuckle curves, seams, profile systems, and reference curves to enhance visibility. For more information, see *Modify Plate Knuckle Curve Display Colors* (on page 939).

Plate Knuckle Properties Dialog Box - Specifies the properties for a plate knuckle. For more information, see *Plate Knuckle Properties Dialog Box* (on page 931).

Profile Knuckle Properties Dialog Box - Specifies the properties for a profile knuckle. For more information, see *Profile Knuckle Properties Dialog Box* (on page 933).



Profile Knuckle - Creates a manual knuckle on a profile system. For more information, see *Profile Knuckle* (on page 935).

Plate Knuckle Properties Dialog Box

Specifies the properties for the selected plate knuckle. The software creates a plate knuckle when the curve of an extruded plate system has an angle with no radius. For more information, see *Linear Extruded Plate System* (on page 156) and *Nonlinear Extruded Plate System* (on page 180).

General Tab (Plate Knuckle Properties Dialog Box) (on page 931)

Relationship Tab (on page 123)

Configuration Tab (on page 124)

General KNU Tab (Plate Knuckle Properties Dialog Box) (on page 933)

General Tab (Plate Knuckle Properties Dialog Box)

Specifies the properties of the plate knuckle.

Name

Specifies the name of the knuckle. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Specifies the naming rule to use to name the knuckle.

Type

Specifies the type of knuckle that you are placing.

Landing Curve Definition Method

Specifies the landing curve definition method for the knuckle.

Parent System

Specifies the parent system for the knuckle. You can define parent systems in the Systems and Specifications task. When the plate system is created, the property values for the parent system are used for the initial plate system property values. When a parent property value changes, the corresponding child property value is updated.

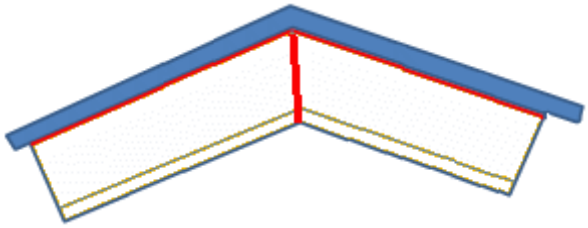
Override

When selected, makes the **Manufacturing Method** box editable.

Manufacturing Method

Select the method for creating the knuckle. This box is read-only unless the **Override** check box is turned on.

- **Bend Knuckle** - The knuckle part of the plate display the bend radius after detailing. Plates and profiles that cross the knuckle include the knuckle bend in their contour after detailing. Manufacturing parts have knuckle marking information.
- **Split Knuckle** - A design seam is placed at the same location as the knuckle on the plate. You must run the *Execute Split* (on page 891) command to split the plate system. After detailing, the plate edges along the knuckle bisect the knuckle angle. Plates that cross the knuckle include the knuckle angle in their contour after detailing. The landing curves of profiles that cross the knuckle follow the knuckle contour. Profiles also receive a knuckle point and are split after detailing.



- **Ignore Knuckle** - No seam or radius is created.

Inner Radius

Specifies the inner radius dimension of the knuckle. The inner radius is generally two times the web thickness. This box is read-only if **Override Rule** is cleared.

Override Radius

When selected, makes the **Inner Radius** box editable.

Molded Radius

Specifies the bend radius to the molded surface. The molded bend radius is automatically calculated based on the inner radius, plate thickness, plate thickness direction, and plate offset if any. This box is read-only.

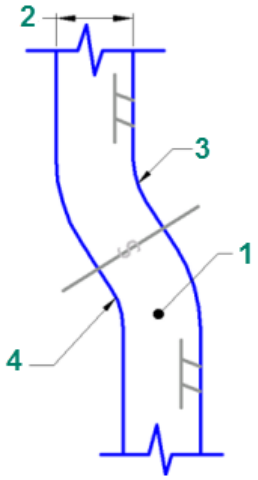
An example potential knuckle radius is shown in the following figure. The inner knuckle's radius is defined as the angle less than 180 degrees at the knuckle.

If the *plate thickness direction* is from *left to right*, the software calculates the following:

Molded radius at the left side = Inner radius at the left side;
Molded radius at the right side of the plate - Inner radius at the right side + plate thickness

If the plate *thickness direction* is from *right to left*, the software calculates the following:

Molded radius at the left side = Inner radius at left side + plate thickness;
Molded radius at the right side of the plate = Inner radius at the right side.



1 - Plate part

2 - Plate thickness

3 - Inner knuckle radius

4 - Outer knuckle radius

General KNU Tab (Plate Knuckle Properties Dialog Box)

The **General** tab displays the properties that were selected by you or automatically determined by the software at creation. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

The properties displayed duplicate those available on other tabs, such as the **Main** tab. You can edit the properties on these tabs.

Profile Knuckle Properties Dialog Box

Specifies the properties for the selected profile knuckle. The software creates a profile knuckle when a sketched profile has an angle with no radius, or when a profile crosses a knuckle on its parent plate system. For more information, see *Stiffener Profile System by Projection* (on page 746).

Profile Knuckle Tab (Profile Knuckle Dialog Box) (on page 933)

Configuration Tab (on page 124)

General Tab (on page 125)

Profile Knuckle Tab (Profile Knuckle Dialog Box)

Specifies the properties of the profile knuckle.

Name - Specifies the name of the knuckle. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule - Select the naming rule to name the knuckle.

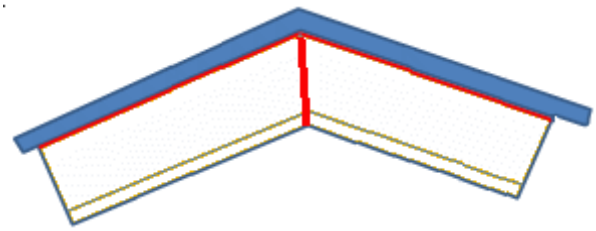
Override Rule - Indicates whether you can manually specify the **Manufacturing Method**. When this option is cleared, the catalog rules determine the behavior of the profile.

Knuckle Properties

Manufacturing Method - Specifies the method for creating the knuckle. This list is only available if the **Override Rule** check box is selected. The availability of methods depends on the profile cross-section type and the plate seam condition. The list is defined by the ProfileKnuckleManufacturingMethod codelist.

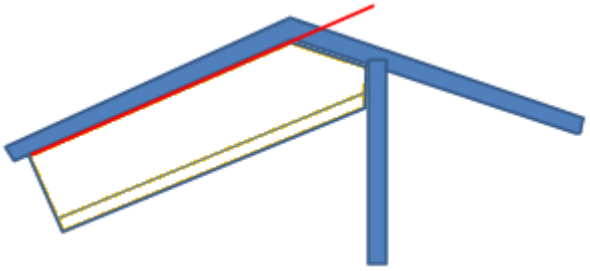
Bend indicates that the knuckled part displays the bend radius after detailing. The Flat Bar cross-section type is supported for this option.

Split indicates that the knuckle point behaves as a design seam. You must run the *Execute Split* (on page 891) command to split the profile system. During detailing, the end cut rules determines the part geometry. The plane bisecting the knuckle angle is passed to the end cut rules. All cross-section types are supported.

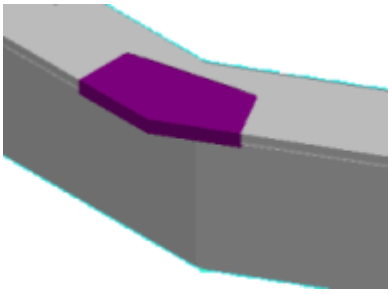


Ignore indicates that no seam or radius is created. This applies to EA, UA, BF and Flat Bar cross-section types. This option is available when the knuckle angle is 2° or less.

Extend indicates that the tangential extension of the profile can extend beyond the profile seam. The software determines part geometry and applicable end cuts. The profile is trimmed to the plate contours after detailing. All cross-section types are supported.



Bend Plus Feature indicates that when a profile is bent about its weak axis, Smart 3D places a feature or features on a profile after detailing. The resulting features that are placed are based on a combination of the profile knuckle rule and the smart occurrence rules. For example, an edge feature and an insert plate feature are placed on a profile that has a concave bend with its flange oriented on the convex side. This applies to UA and EA cross-section types with heights of 200 mm or more and knuckle angles of 90° or less.



Inner Radius - Specifies the inner radius dimension of the knuckle. The inner radius is generally two times the web thickness. This box is read-only if **Override Rule** is cleared.

Outer Radius - Specifies the outer radius dimension of the knuckle. The outer radius is generally the inner radius plus the flange length. This box is read-only if **Override Rule** is cleared.

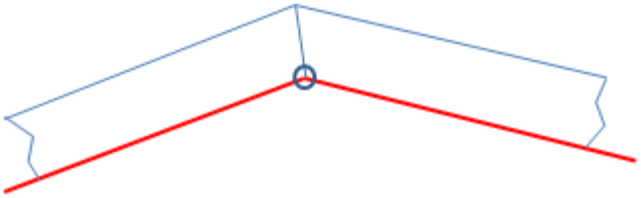
Knuckle Angle - Displays the angle of the knuckle bend.

Related Plate Knuckle - Displays the name of the associated plate knuckle.

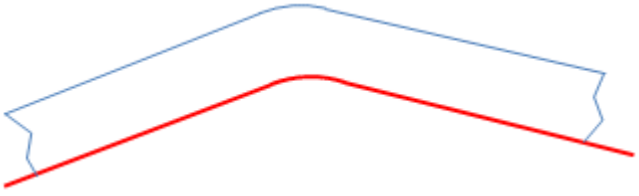
Profile Knuckle

Creates a manual knuckle on the selected profile system. Use this command when a profile crosses a change in the surface direction of in a plate, but the plate has a radius instead of a knuckle and a knuckle is not automatically created for the profile. With **Profile Knuckle**, you can select the radius and create a knuckle at the mid-point of the radius.

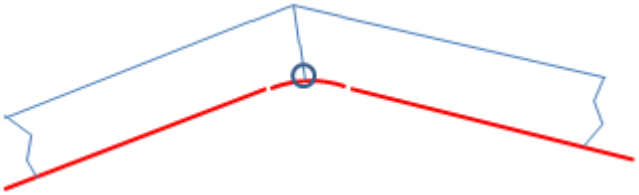
A profile knuckle is automatically created on a knuckled plate:



A profile knuckle is not created when the plate transition has a radius:



Profile Knuckle creates a manual profile knuckle at the midpoint of the radius:



Profile Knuckle Ribbon

Profile Knuckle Properties - Displays the **Manual Profile Knuckle Properties** dialog box, which you use to view and modify the properties of the knuckle. For more information, see *Profile Knuckle Tab (Manual Profile Knuckle Dialog Box)* (on page 938).

Select Profile System - Specifies the profile system on which you want to place a manual knuckle.

Reference Curve - Specifies the reference curve, which you want to use to place a manual knuckle. You can select only one reference curve.

Generate Knuckle Point through GC - Allows creation of a geometry construction point to use as the reference to place the manual knuckle. You create the geometry directly in a graphic view by using the **Geometric Construction Palette** dialog box, the **Geometric Construction** ribbon, and the **Geometric Construction Explorer**. This option is the 3D equivalent to Sketch 2D. For more information, see Geometric Construction Palette Dialog Box and *Geometric Construction Explorer* (on page 1214).

Finish - Places the manual knuckle using the reference you chose.

Manufacturing Method - Specifies the default manufacturing method for the manual knuckle. Select one of the following options:

- **Split** - The knuckle point behaves as a design seam. You must run **Execute Split** to split the profile system. During detailing, the end cut rules determines the part geometry. The plane bisecting the knuckle angle is passed to the end cut rules. For more information, see *Execute Split* (on page 891).
- **Ignore** - The profile part is not affected. The knuckle point is ignored and no knuckle is created.
- **Extend** - Allows the tangential extension of the profile beyond the profile seam. The Structural Detailing task determines part geometry and applicable end cuts.
- **Split and Extend** - Allows the profile system to split at the knuckle, and also at the tangential extension of the part beyond the knuckle.

AutoSplit - Splits the profile system when **Finish** is clicked.

NOTE If this option is not selected, click **Execute Split** to split the profile system. For more information, see *Split a profile or member system* (on page 893).

Knuckle Best Practices **(on page 56)**

What do you want to do?

- *Place a manual profile knuckle by GC point* (on page 937)
- *Place a manual profile knuckle by reference curve* (on page 937)
- *Edit a manual profile knuckle* (on page 938)
- *Delete a profile knuckle* (on page 938)

Place a manual profile knuckle by GC point

1. Click **Profile Knuckle** on the vertical toolbar.
2. Select a profile system crossing a curved plate transition.
3. Click **GC Point**.

   *The **Geometric Construction Palette**, **Geometric Construction** ribbon, and the **Geometric Construction Explorer** display.*
4. In the **Geometric Construction** box of the ribbon or the **Curves** tab of the **Geometric Construction Palette**, select **EdgesFromCurveOrSurface**.
5. In the graphic view or the Workspace Explorer, select the profile system.
6. Click **Continue**.
7. In the **Geometric Construction** box of the ribbon or the **Points** tab of the **Geometric Construction Palette**, select **PointAtCurveMiddle** to specify the midpoint of the arc to place the profile knuckle.
8. In the **Geometric Construction Explorer**, select the needed arc on the profile landing curve. Select **[Not Named]** elements under **EdgesFormCurveOrSurf** until the correct curve is highlighted in the graphic view.



9. Click **Continue**, and then click **Close**.
10. Select the manufacturing method.
11. Click **Finish**.

    A knuckle point is added to the profile at the midpoint of the arc.

Place a manual profile knuckle by reference curve

1. Click **Profile Knuckle** on the vertical toolbar.
2. Select the root profile system.
3. Click **Reference Curve**.

4. Select a reference curve.
5. Click **Finish**.

Edit a manual profile knuckle

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter** box, select **RefCurve On Surface & Knuckles**.
3. In a graphic view or on the Workspace Explorer, select the manual knuckle to edit.
4. Edit the knuckle as needed.
5. Click **Finish**.

Delete a profile knuckle

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter** box, select **RefCurve On Surface & Knuckles**.
3. In a graphic view or on the Workspace Explorer, select the manual knuckle to delete.
4. Click **Edit** > **Delete**.

Manual Profile Knuckle Properties Dialog Box

Specifies the properties for the profile knuckle.

Profile Knuckle Tab (Manual Profile Knuckle Dialog Box) (on page 938)

Profile Knuckle Tab (Manual Profile Knuckle Dialog Box)

Specifies the properties for the manual profile knuckle that you are placing or editing.

Name - Specifies the name of the knuckle. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see "Using Global Workshare" in the Global Workshare Guide.

Rule - Specifies the naming rule to name the knuckle.

Knuckle Properties

Override Rule - Allows you to manually specify the manufacturing method. If this option is cleared, the rules determine the method.

Manufacturing Method - Specifies the method for creating the manual profile knuckle. This option is only available if **Override Rule** is selected. Select one of the following options:

- **Split** - The knuckle point behaves as a design seam. You must run **Execute Split** to split the profile system. During detailing, the end cut rules determines the part geometry. The plane bisecting the knuckle angle is passed to the end cut rules. For more information, see *Execute Split* (on page 891).
- **Ignore** - The profile part is not affected. The knuckle point is ignored and no knuckle is created.

- **Extend** - Allows the tangential extension of the profile beyond the profile seam. The Structural Detailing task determines part geometry and applicable end cuts.
- **Split and Extend** - Allows the profile system to split at the knuckle, and also at the tangential extension of the part beyond the knuckle.

NOTE You cannot modify the properties for **Manufacturing Method** when creating a new profile knuckle.

Inner Radius - Specify the inner radius dimension of the knuckle. This box is read-only if **Override Rule** is cleared.

Outer Radius - Specify the outer radius dimension of the knuckle. This box is read-only if **Override Rule** is cleared.

Knuckle Angle - Displays the angle of the knuckle bend.

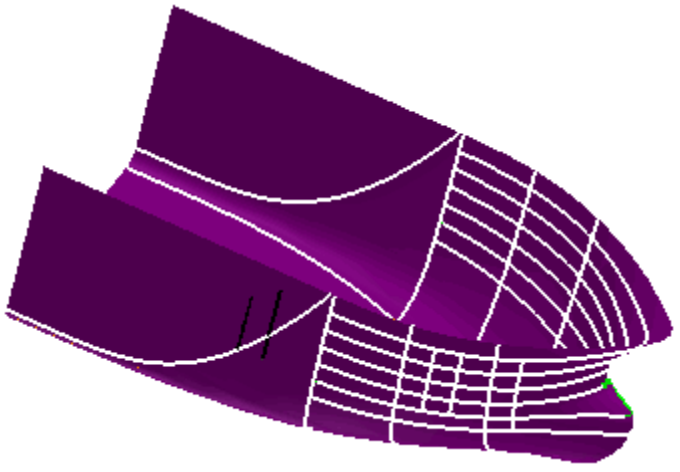
Related Plate Knuckle - Displays the name of the associated plate knuckle. A plate knuckle is associated with a profile knuckle if you select it as a reference object.

Modify Plate Knuckle Curve Display Colors

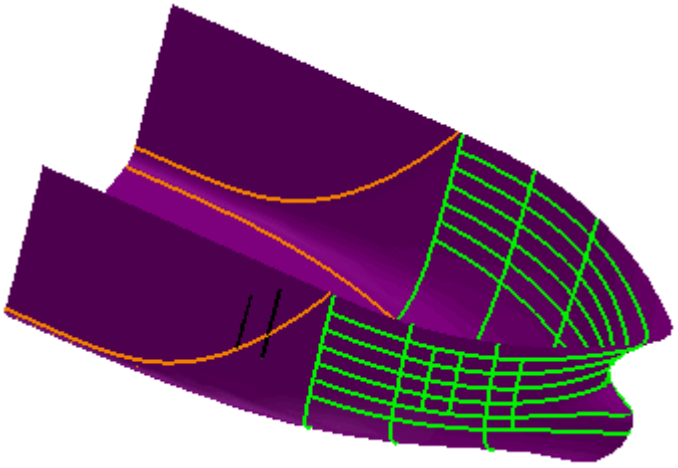
Use this procedure to create style rules to customize the display colors of landing curves. You can use the procedure to set different colors for the landing curves of:

- Seams
- Profile Systems
- Reference Curves
- Plate Knuckle Curves

For example, before creating style rules, reference curves and design seams are white:



After creating separate style rules for reference curves and design seams, they appear in different colors:



Use the following steps to customize curve display colors.

Select the View Aspect

1. Click **Format** >**View**.

   *The **Format View** dialog box opens.*

2. In the **Selected Aspects** box, click **Reference Geometry**.

   NOTE By default, **Simple physical** is also selected.

3. Click **OK**.

Create a Style Rule

1. Click **Format** > **Surface Style Rules**.

   *The **Surface Style Rules** dialog box opens.*

2. Click **New**.

   *The **Surface Style Rule Properties** dialog box opens*

3. Type a name for the new surface style rule in the **Rule name** box.
4. In the **Filter** list, click **More** to create a new filter.

   *The **Select Filter** dialog box opens.*

5. Click **New Filter**.

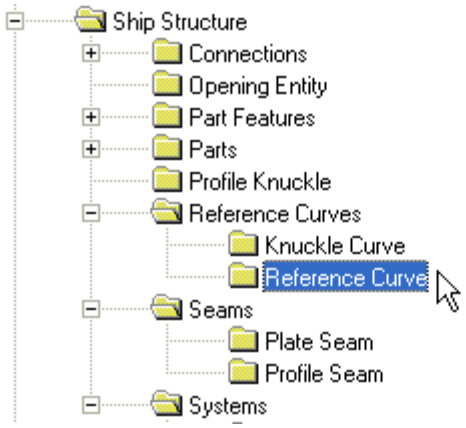
   *The **New Filter Properties** dialog box opens.*

   NOTE If you are an administrator, you can first select **Ship Filters**. The new filter is then available to all users.

6. On the **Object Type** tab, expand the hierarchy **Ship Structure**, and select one of the following:

   - Ship Structure/ Reference Curves/**Reference Curve**

- Ship Structure/ Reference Curves/**Knuckle Curve**
- Ship Structure/Seams/**Plate Seam**
- Ship Structure/Systems/Profile Systems/**Stiffener System**



7. Click **OK**.
8. Click **OK** in the **Select Filter** dialog box.
9. In the **Surface Style Rule Properties** dialog box, select a color in the **Style applied** box.
10. Click **OK**.

    *The new rule appears in the **Style rule library** list.*

Add the Rule to the Workspace

1. Select the rule in the **Style rule library** list.
2. Click **Add**.
3. Click **OK**.

See Also

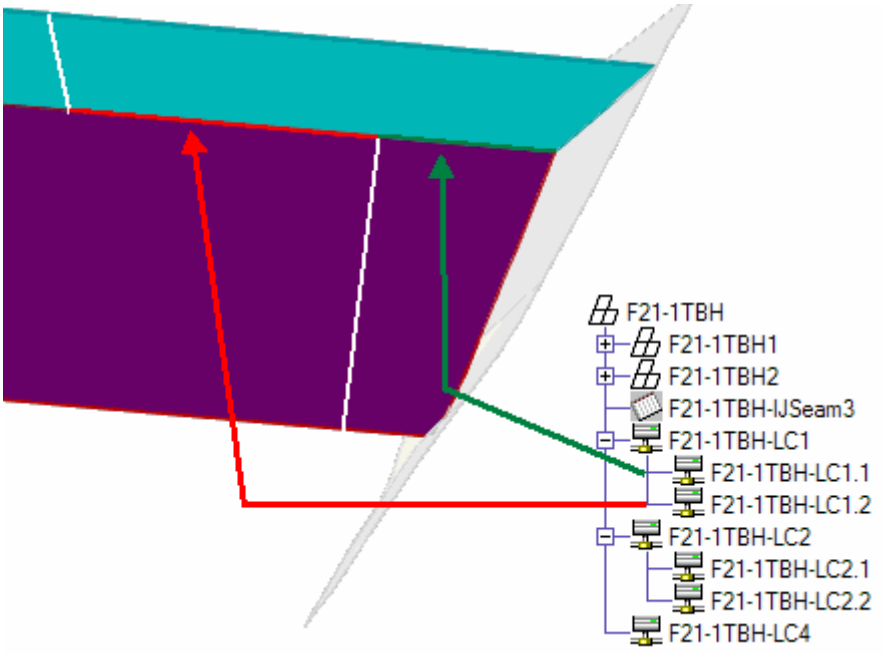
In the Common User's Guide:
Format View
Format a View
Surface Style Rules
Create a Surface Style Rule
Create a New Filter
Add a Surface Style Rule to the Workspace

SECTION 12

Placing Logical Connections

Logical connections are created automatically as children of plate and profile systems where they are bounded to other systems. When placing design seams and splitting the seams, logical connections are also created automatically at the seams for the newly created child systems or at the intersections between systems. In some cases, you can manually create a logical connection, such as between two edge-to-edge plate systems or between a profile system and a plate system that are aligned end-to-edge.

The top level logical connection is called the root logical connection. Every root logical connection has at least one leaf (or child) logical connection. If multiple leaf logical connections exist, then the parent system is bounded to or intersects multiple child systems along the connection, as shown in this example:



When two plate systems or a plate system and a profile system intersect or are bounded to each other, you can invert the logical connection to change the way the systems are connected. You can also split the logical connection between the plate systems.

You can manually place logical connections, such as between edge-to-edge plate systems, or between members and plates. For more information, see *Logical Connection* (on page 943).

Logical connections contain the connection properties between systems. These properties are used to create assembly connections, physical connections, and construction feature objects in the Structural Detailing task, and to generate manufacturing data in the Structural Manufacturing task.

See Also

Creating Seams (on page 888)
Edit Intersection (on page 948)

Logical Connection

The **Logical Connection** command creates a relationship between two systems to identify that the two systems are connected. Use this command when you have placed the two systems separately, but they coincide such as described in the following situations:

- A plate system edge against another plate system edge. This situation can happen when you use the same grid plane as a boundary for both plate systems during placement. You can connect plate systems of the same type (planar, imported, ruled, linear extruded, nonlinear extruded, revolved, or child) or different types. An example of this situation is a cambered deck where you want the seams at the deck to be knuckles. If the deck is created as separate planar plate systems sharing grid plane boundaries, use this command to connect the separate plate systems.
- A plate system edge against a stiffener profile (or linear beam profile) end. This situation can happen when the ends of edge reinforcements connect to the free edge of plate system openings or sketched boundaries.
- A profile end against profile end. This situation can happen when you have two profile systems on adjacent plate systems.
- A member to a plate system. This situation can happen when members are created first, and plates are added afterwards.
- Two lapped plates, or between a plate and a member. This situation can happen when the surface of one flat plate is adjacent to the surface of another flat plate, or when a member is adjacent to a plate.

Plate Knuckles

If the plate system has an angle with no radius on the extrusion curve, the software creates a plate knuckle object. You can select this object graphically, or in the **Workspace Explorer**. For more information, see *Plate Knuckle Properties Dialog Box* (on page 931).

Profile Knuckles

If the plate system associated with the profile has an angle with no radius on the extrusion curve, the software creates a profile knuckle object. You can select this object graphically, or in the **Workspace Explorer**. For more information, see *Profile Knuckle Properties Dialog Box* (on page 933).

Logical Connection Ribbon

Logical Connection Properties - Opens the **Logical Connection Properties** dialog box, which you use to define connection properties. For more information, see *Logical Connection Properties Dialog Box* (on page 945).

Finish - Places a logical connection between the objects that you have selected.

Reject - Clears all selected objects.

Accept - Accepts all selected objects.

Connection Type to Create - Specifies the type of logical connection to be created. Select one of the following in the list: **Member Lapped to Plate**, **Plate Lapped to Plate, Profile End to Profile End**, **Plate Edge to Plate Edge**, **Any System Connected to Any System**, or **Any System Lapped to Any System**. The connection type that you select determines which pair of **Select Set** controls display on the ribbon. See *Select Set Controls* below.

Parent - Specifies the parent object for the logical connection. Select **Use Select Set 1 Object as Parent** or **Use Select Set 2 Object as Parent**.

Name - Displays the name of the logical connection.

Lapped - Designates that the logical connection is between two lapped plates. This option is checked by default when the logical connection is between a member and a plate.

Select Set Controls

The following table shows the controls that display on the **Logical Connection** ribbon when a selection is made in the **Connection Type** list. Use these controls to help identify the objects that you can select to create the required logical connection.

Connection Type	Select Set 1	Select Set 2
Member Lapped to Plate	**Plate System** - Select a plate system to connect.	**Standard Member** - Select one or more standard members to connect.
Plate Edge to Plate Edge	**Plate System** - Select one or more plate systems to connect.	**Plate System** - Select one or more plate systems to connect.
Profile End to Profile End	**Profile System** - Select one or more profile systems to connect.	**Profile System** - Select one or more profile systems to connect.
Plate Lapped to Plate	**Plate System** - Select a plate system to connect.	**Plate System** - Select the other plate system to connect
Any System Connected to Any System	**First System** - Select any system to connect.	**Second System** - Select the other system to connect.
Any System Lapped to Any System	**First System** - Select any system to connect.	**Second System** - Select the other system to connect.

What do you want to do?

- *Connect systems using a logical connection* (on page 945)
- *Remove logical connections* (on page 945)

Connect systems using a logical connection

1. Click **Logical Connection** on the vertical toolbar.
2. In the **Connection Type to Create** list, select the type of connection to create.
3. In the **Parent** list, specify the select set to be used as the parent.
4. Create the first select set, and click **Accept**.

   TIPS

   - The select set can consist of one or more systems. To select multiple objects, press the CTRL key while clicking each required member of the select set. Alternatively, drag the mouse to fence the required select set objects.
   - Click **Reject** to clear the select set of all selected objects.
5. Create the second select set, and click **Accept**.
6. Repeat the previous steps as needed to create select sets for as many logical connections as necessary.
7. Click **Finish**.

   The software creates the logical connections. Logical connections are highlighted in red in the graphic view.

NOTE The logical connections remain highlighted in red in the graphic view until **Finish** is clicked again or until the selection in the **Connection Type** list changes.

Remove logical connections

1. Click **Select** on the vertical toolbar.
2. Select **Connections** from the **Locate Filter**.
3. Select the logical connection to remove.
4. Click **Edit > Delete**.

Logical Connection Properties Dialog Box

Specifies the properties for logical connections.

Main Tab (Logical Connections Properties Dialog Box) (on page 945)

Relationships Tab (Logical Connections Properties Dialog Box) (on page 947)

Configuration Tab (on page 124)

General Tab (on page 125)

Main Tab (Logical Connections Properties Dialog Box)

Specifies the main properties for the logical connection.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare

name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Specifies the naming rule to use to name the logical connection.

- Select **StdHierarchyChildNamingRule** to use the following syntax: <*Parent Name*>-LC<*Index Number*>. For example, D0-1DCK-1-LC101, where D0 is the reference plane, 1DCK is an index number appended by the plate type (Deck in this example), 1 is the Workshare Location ID, and LC101 is "Logical Connection" appended by an index number.
- Select **User Defined** to name the logical connection yourself by using the appropriate box.

Parent System

Displays the name of the parent system of the logical connection.

Type

Displays the object type, which in this case is **Logical Connection**.

Optional Corner Trim Property

NOTE Your administrator must bulkload the following files before the **Corner Trim** displays:

Bulkload into the site and site schema databases:

- *[Product Folder]*\ShipCatalogData\BulkLoad\DataFiles\ProjectDBFiles\StructureCustomClasses.xls
- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimCodelist.xls
- Bulkload into the catalog and catalog schema databases:
- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\SM_SquareEdgeTrimRules.xls
- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimCodelist.xls
- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimAttribute.xls

Corner Trim

Indicates whether plates bounded by plate connections or edge reinforcement connections use the square edge trim.

The available options are:

- **Flush (Flush Neat Trim)** - This value uses the normal trim behavior for attached plates or for plates bounded by edge reinforcement when detailing. This is the default value.
- **Open (Open Square Trim)** - This value uses the square edge trim behavior when detailing the attached plates or when detailing the plates bounded by edge reinforcement, as shown below.

Plate bounded by plate:

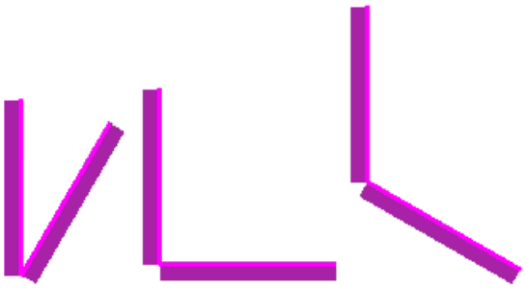
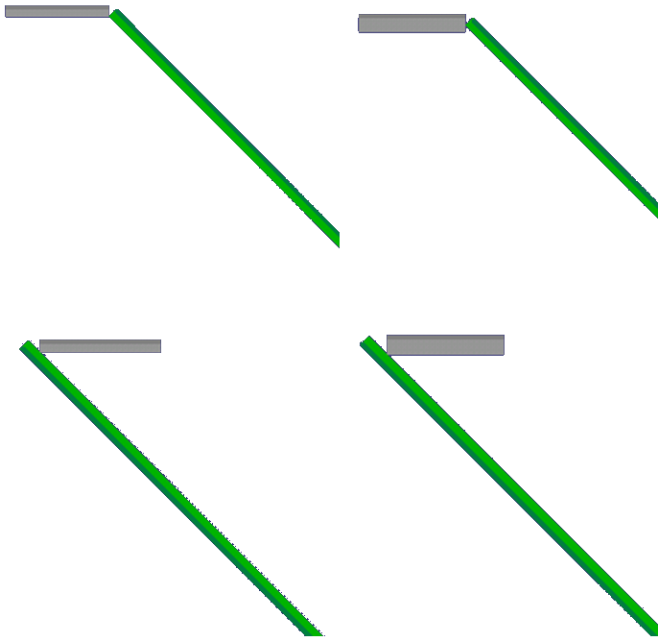


Plate bounded by edge reinforcement:



Relationships Tab (Logical Connections Properties Dialog Box)

Displays a list of objects related to the logical connection. Select **General** or **Logical Connection** as the relationship category from the list on the right.

List color - Specifies the display color in the graphic view for all items in the list.

Item color - Specifies the display color in the graphic view for a selected item in the list.

Name - Displays the name of the related object.

Type - Displays the type of related object.

Go To - Displays the properties of the selected object.

Edit Intersection

When two plate systems or a plate system and a profile system intersect or are bounded to each other, you can use the **Edit Intersection** command to change the way the systems are connected. The **Edit Intersection** ribbon is available when you select any root logical connections between two plate systems.

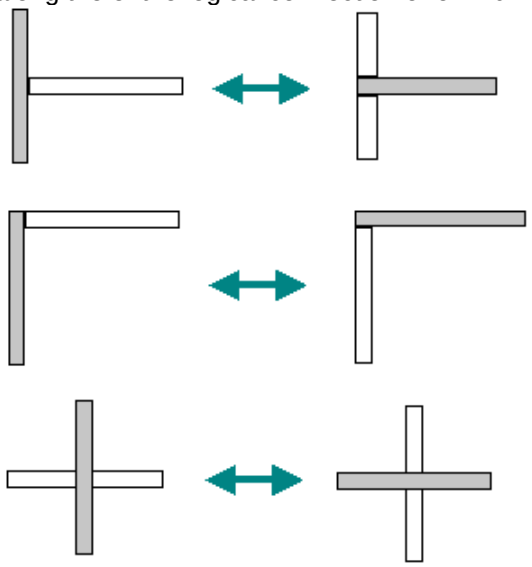
NOTE However, the ribbon is not available for logical connections between a plate and a profile. Only the **Invert** option is available from the short-cut menu.

You can edit logical connections as follows:

- Split a logical connection into regions.

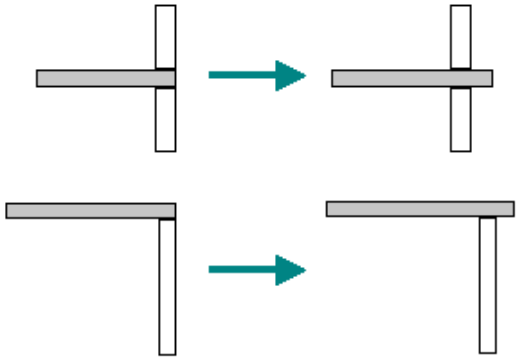
  NOTE The software does not support splitting the logical connection between a profile and a plate.

- Invert the connection relationship (continuous versus intercostal) between each system, along the entire logical connection or on individual regions.



- Offset the edge of one system beyond the other, along the entire logical connection or on individual regions.

  NOTE The offset does not apply to the logical connection between a profile and a plate.



Edit Intersection Ribbon

NOTES

- The **Edit Intersection** ribbon is available when you select any root logical connections between two plate systems. However, the ribbon is not available for logical connections between a plate and a profile. Only the **Invert** option is available from the short-cut menu.
- The software does not support splitting the logical connection between a profile and a plate.

Logical Connection Properties - Opens the **Logical Connection Properties** dialog box, which you use to modify connection properties. For more information, see *Logical Connection Properties Dialog Box* (on page 945).

Define Region - Select one or more objects that intersect the logical connection to define region boundaries. You can select objects such as reference planes, reference curves, seams, and points. If no objects are selected, then the logical connection has one region.

Select and Edit Region - Select a region to edit.

Finish - Adds the regions and offsets to the logical connection.

Cancel - Clears all selected objects.

Accept - Accepts all selected objects.

Invert - Reverses the connection relationship (continuous versus intercostal) between the systems in the selected region. An arrow in the direction of the continuous system appears in a graphic view.

NOTES

- The **Invert** option can simultaneously invert multiple logical connections when each connection has only one region. If any of the selected logical connections have multiple regions, the invert operation fails with an error message.
- If a single logical connection is selected to be inverted and it has only one region, the region is automatically selected for the invert operation.
- For logical connections between a plate and a profile, only the **Invert** option is available from the short-cut menu.

Offset - Specifies a value to extend the continuous system beyond the intercostal system. Available when at least one of the systems is bounded to the other.

NOTE The offset does not apply to the logical connection between a profile and a plate.

What do you want to do?

- *Modify a logical connection* (on page 950)
- *Invert multiple logical connections* (on page 950)
- *Invert the logical connection between a plate and a profile* (on page 951)

Modify a logical connection

NOTE The software does not support splitting the logical connection between a profile and a plate.

1. Click **Select** on the vertical toolbar.
2. Select **Connections** in the **Locate Filter**.
3. Select the logical connection to modify.

   *The **Edit Intersection** ribbon appears.*
4. If you need to split the logical connection into more than one region, click **Define Region**.
5. Select one or more grid planes, plate systems, profile landing curves, or seams that intersect the logical connection.
6. Click **Select and Edit Region**.

   A red circle appears at each region boundary.
7. Select a region to modify.

   A red arrow appears in the plane of the continuous system.
8. If needed, click **Invert**.

   The red arrow moves to the other system, which is now continuous for the selected region.

   TIP You can also right-click the logical connection in a graphic view or in the **Workspace Explorer** and click **Invert** from the short-cut menu.

   NOTE If the logical connection to be inverted has only one region, the region is automatically selected for the invert operation.
9. If needed, type a value for **Offset**.
10. Click **Finish**.

    NOTE **Invert** and **Offset** changes made to the logical connection are not visible in a graphic view until the systems have been detailed in the **Structural Detailing** task.

Invert multiple logical connections

NOTE The software does not support splitting the logical connection between a profile and a plate.

1. Click **Select** on the vertical toolbar.
2. Select **Connections** in the **Locate Filter**.
3. Select the logical connections that you want to invert.

   *The **Edit Intersection** ribbon appears. If all the selected logical connections have only one region, the region is automatically selected for the invert operation, a red circle appears at each region boundary, and a red arrow appears in the plane of each continuous system. If any of the logical connections have multiple regions, you can only modify their logical connection properties. Only **Logical Connection Properties** is available on the ribbon bar.*
4. Click **Invert**.

For each logical connection, the red arrow moves to the other system, which is now continuous for the region.

TIP You can also right-click the logical connections in a graphic view or in the Workspace Explorer and click **Invert** from the short-cut menu.

NOTE The logical connections to be inverted simultaneously must have only one region. If any of the logical connections have multiple regions, the operation fails with an error message.

5. If needed, type a value for **Offset**. This value is applied to all selected logical connections.
6. Click **Finish**.

NOTE **Invert** and **Offset** changes made to the logical connections are not visible in a graphic view until the systems have been detailed in the Structural Detailing task.

Invert the logical connection between a plate and a profile

NOTE The software does not support splitting the logical connection between a profile and a plate.

1. Click **Select** on the vertical toolbar.
2. Select **Connections** in the **Locate Filter**.
3. In a graphic view or the Workspace Explorer, select the logical connections that you want to invert.

   If all the selected logical connections have only one region, the region is automatically selected for the invert operation, a red circle appears at each region boundary, and a red arrow appears in the plane of each continuous system.

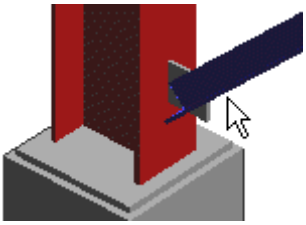
   NOTE The logical connections to be inverted simultaneously must have only one region. If any of the logical connections have multiple regions, the operation fails with an error message.
4. Right-click the logical connections and click **Invert** from the short-cut menu.

   The software immediately inverts the connections. For each logical connection, the red arrow moves to the other system, which is now continuous for the region.

NOTE Invert changes made to the logical connections are not visible in a graphic view until the systems have been detailed in the Structural Detailing task.

SECTION 13

Place Assembly Connection



Places an assembly connection or free end cut at the selected frame connection. Assembly connections define the necessary trimming between member parts and provide for the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections also control cutbacks, copes, notches, bolt holes, weld preparations, and slots.

This command places member assembly connections or a free end cut on the member. The software automatically places slab assembly connections when you place the slab.

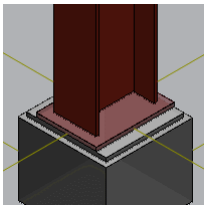
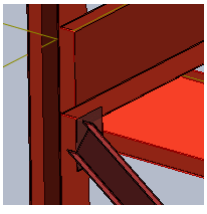
Member Assembly Connections

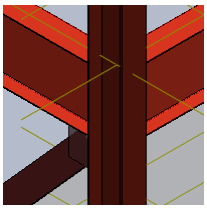
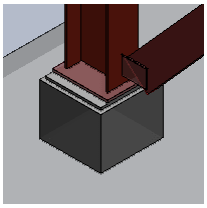
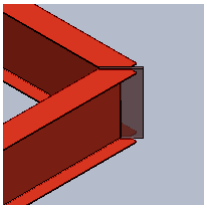
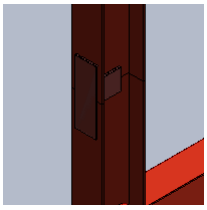
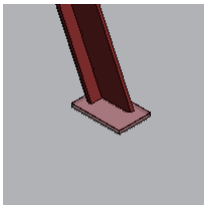
Member assembly connections are similar to frame connections, but define the necessary trimming between member parts and the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections control member features including cutbacks, copes, notches, bolt holes, and slots. Whether or not features are placed depends on the member assembly connection type and the geometry of the connection between the members.

There are several basic assembly connections delivered with the software. You can create your own assembly connections by editing the **StructAssemblyConnections.xls** workbook, and then bulk loading the workbook. For more information on creating your own assembly connections, see the Structure Reference Data Guide.

NOTE For very complex nodal connections, instead of writing a custom assembly connection, you might want to try *Trim Members* in the Structure task.

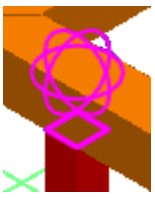
The following general assembly connections are delivered with the software:

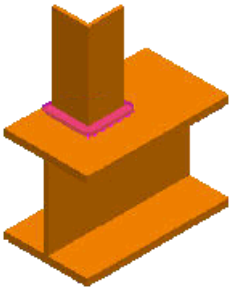
	The base plate assembly connection places a plate at the end of an unsupported member. This assembly connection requires an unsupported frame connection on one member. For example, use this assembly connection to place a base plate at the bottom of a column.
	The corner gusset plate assembly connection connects a vertical brace to a beam and column intersection using a gusset plate. This assembly connection requires a frame connection with three members, such as vertical corner brace.

	The fitted assembly connection connects two members. This assembly connection requires a frame connection with two members, such as axis, seated, or flush. Examples of this connection include a beam framing into a column or a beam framing into another beam.
	The gusset plate assembly connection connects a vertical or horizontal brace to a beam or a vertical brace to a column using a gusset plate. This assembly connection requires a frame connection with two members, such as axis.
	The miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection requires a frame connection with two members, such as axis. In addition, the members must be end connected.
	The splice assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The general surface assembly connection connects a member end with a nonmember surface such as a slab, a deck, or a hull plate. The member is cut to surface and a base plate is placed on the member end.

End Cut Assembly Connections

The following end cut assembly connections are delivered with the software:

	The end cut along assembly connection cuts the end of one member to the sides or bottom of another member along the second member's length.

	The end cut bearing plate assembly connection connects the end of a member along the axis of another member. This is typically used to attach the end of a member to the flange of the supporting member. The bearing plate connection works at any angle and provides circular, rectangular, and triangular bearing plates depending on the supported member's shape.
	The end cut miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection required a frame connection with two members, such as axis. In addition, the members must be end connected.
	The end cut split assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The end cut split long box assembly connection is used when the supported and supporting members are perpendicular to each other. The supporting member is cut square to the supported member while an item from the Axis category is placed on the supported member.
	The end cut split seam angle assembly connection is used when the supported and supporting members are: ▪ member axes are collinear to each other ▪ have identical cross-section type and size ▪ have identical cardinal point and rotation values
	The end cut split short box assembly connection is used when the supported and supporting members are perpendicular to each other. The supported member is cut square to the supporting member while an item from the Axis category is placed on the supporting member.

End Cut Assembly Naming Conventions

Each end cut assembly connection name consists of three parts:
[EndCutCase]_[WebCutType]_[FlangeCutType]

where:

- [EndCutCase]
  - **Axis** - Supported member is bounded by the axis curve of the supporting member
  - **LongBox** - End-to-end perpendicular supported member is extended, and an axis end cut is applied to the supporting member
  - **ShortBox** - End-to-end perpendicular supporting member is extended, and an axis end cut is applied to the supported member

- **Miter** - End-to-end non-collinear
- **Split** - End-to-end collinear
- **SeamAngle** - End-to-end collinear where one end is extended

- [WebCutType]

  The WebCutType is used to type of web cut that is to be applied at the top of the web and at the bottom of the web. The following combinations are available (W indicates a straight cut, C indicates a Cope cut, and S indicates a Snipe cut):

  - W#W#
  - W#C#
  - W#S#
  - C#C#
  - C#S#
  - C#W#
  - S#S#
  - S#C#
  - S#W#

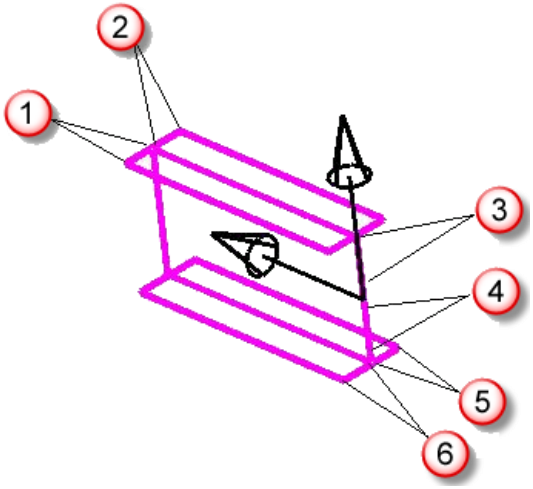
- [FlangeCutType]

  The FlangeCutType is used to type of flange cut that is to be applied at the left flange section and at the right flange section. The following combinations are available (W indicates a straight cut, C indicates a cope cut, and S indicates a snipe cut):

  - W#W#
  - W#S#
  - S#S#
  - S#W#

You cannot apply different flange cuts when the supported member has both a top flange and a bottom flange. In these cases, the same flange cut type is applied to both the top and bottom flange.

You cannot apply different web cuts or flange cuts in the end-to-end cases. The end-to-end cases place a web cut and flange cut on both the supported member and the supporting member. For these cases, the same web cut and flange cut are applied to both the supported member and the supporting member.

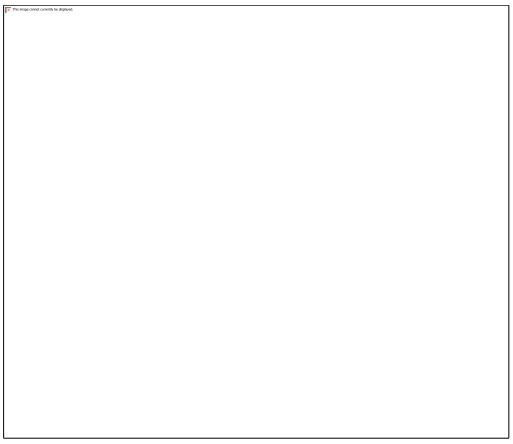


1 - Top left flange
2 - Top right flange
3 - Web top
4 - Web bottom
5 - Bottom right flange
6 - Bottom left flange

See *Members* in the Structure task for important related information.

Locating Assembly Connections

Assembly connections do not display in the model. However, if you set the **Locate Filter** to **Member Assembly Connections** in the Structure task, you can locate and select assembly connections for review and editing. Assembly connections are located at the ends of member parts and appear as circles when you move the cursor over them. Any assembly components, such as gusset plates, associated with the assembly connection also highlight.



When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties. You cannot measure distances using the assembly connection graphic as an end point when using the **Tools > Measure** command.

Place Assembly Connection Ribbon

Specifies the properties for the assembly connection that you are placing or editing.

Assembly Connection Properties

Activates the **Assembly Connection Properties** dialog box. You can use this dialog box to specify additional properties that you cannot set on the ribbon. For more information, see

Assembly Connection Properties Dialog Box (on page 962).

Select Member/Connection

Activated automatically by the software so that you can select the frame connection or the member for which you want to place assembly connections.

Finish

Click to place the assembly connection the model.

Cancel

Rejects the selected object.

Accept

Confirms that the selected members, is the member to place assembly connections for. The software displays in tentative mode the results of the assembly connection.

Condition

Specifies how you want the software to handle existing assembly connections when you try to place a new assembly connection at the same location. Select **Retain existing** to keep the existing assembly connection. Select **Update existing** to replace the existing assembly connection with the new assembly connection.

By Rule

Select to allow the software to select the assembly connection to use based on the selected member parts and their orientation to each other.

Type

Select the assembly connection type to use. If you select **By Rule**, the software determines the correct assembly connection to use based on the geometry between the member parts, and this option is unavailable. If you select **More**, all available assembly connections display from which you can select the assembly connection to use. For more information about assembly connections, see Member Assembly Connections.

Member Part Parent

Select to make the member part the parent object of the assembly connection. Clear this option to specify a generic system, using the **System** box, to be the assembly connection parent. Assembly connections created using member parts as their parents can be edited to have a system parent later if needed.

System

Select the system to which the assembly connection belongs. You can define new systems in the Systems and Specifications task. This option is unavailable if you select **Member Part Parent**. Assembly connections created using a system as their parent can be edited to have a member part as their parent later if needed.

Name

Specify the name of the assembly connection.

Modify Auxiliary Parts Ribbon

Displays the options that you use to add auxiliary parts to the selected assembly connection.



Properties

Activates the **Assembly Connection Properties Dialog Box**, which you use to view and modify the properties of the assembly connection before the assembly connection is committed to the database. For more information, see *Assembly Connection Properties Dialog Box* (on page 962).

Assembly Connection Smart Item

Displays valid smart items for the selected assembly connection.



Add/Delete Auxiliary Parts

Select the parts that can be connected to the selected assembly connection and do not already have an existing assembly connection. Parts that do not have common geometry with the selected assembly connection cannot be selected.



Accept

Adds parts from selected parts to the assembly connection and saves the changes to the database.



Reject

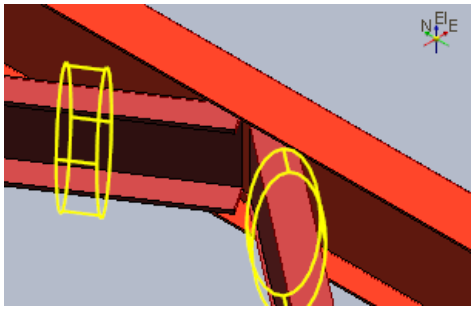
Removes any selected parts from the auxiliary parts list. Only selections made after the last commit are removed.

What do you want to do?

- *Place an assembly connection* (on page 959)
- *Edit assembly connection properties* (on page 960)
- *Delete an assembly connection* (on page 960)
- *Add auxiliary parts to an assembly connection* (on page 960)
- *Edit assembly connection parts* (on page 961)
- *Place free end cuts* (on page 961)
- *Edit free end cut properties* (on page 961)
- *Delete free end cut* (on page 961)

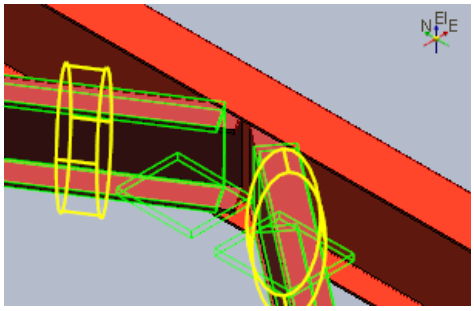
Place an assembly connection

1. Click **Place Assembly Connection** on the vertical toolbar.
2. Select the **By Rule** option if you want the software to automatically select the type of assembly connection to place. Clear the **By Rule** option to select the assembly connection type yourself.
3. Select the frame connection nearest the member end to which to apply the assembly connection.

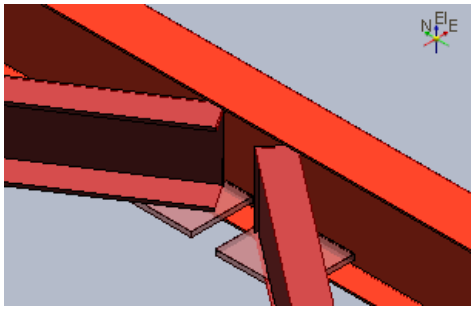


4. If you are selecting the assembly connection type yourself, use the **Type** option.
5. Click **Accept** ✓.

   The software automatically selects the assembly connection and displays the results.



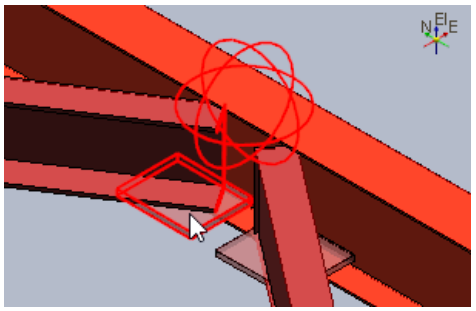
6. Click **Finish**.



NOTE For more information about the types of assembly connections, see Member Assembly Connections.

Edit assembly connection properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter**.
3. Select the assembly connection to edit.



4. Click **Edit > Properties**.
5. Edit the assembly connection properties as needed. For more information, see *Assembly Connection Properties Dialog Box* (on page 962).

Delete an assembly connection

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter**.
3. Select the assembly connection to delete.
4. Click **Delete**.

Add auxiliary parts to an assembly connection

This command supports adding parts to an existing assembly connection by selecting an adjacent plate, profile or member parts that have overlapping geometry.

1. Select an existing assembly connection.
2. Click **Select Auxiliary Parts** on the ribbon bar.

   NOTE The **Select Auxiliary Parts** is enabled for any assembly connection. You can only add valid auxiliary parts to an assembly connection.

3. Click **Accept** to add all intersecting parts on the selected parts to the assembly connection.

Edit assembly connection parts

1. Select an assembly connection, and click **Select Auxiliary Parts** on the ribbon bar.
2. Select the parts not already connected to the selected assembly connection that have intersecting geometry.
3. To remove a single part, re-select it from the graphic view or **Workspace Explorer**.
4. To remove the auxiliary parts added in the current command execution, click **Reject** on the ribbon bar.
5. Click **Accept** to add all intersecting parts on the selected parts to the assembly connection.

Place free end cuts

1. Click **Place Assembly Connection** on the vertical toolbar.
2. Select the unsupported frame connection nearest the member end to which you want to apply the free end cuts.

   If you select a member part, the software automatically places free end cuts at the unsupported frame connections.
3. Click **Accept.**

   The software automatically selects the free end cuts and displays the results.
4. Click **Finish**.

Edit free end cut properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter** box.
3. Select the free end cut to edit.
4. Click **Edit > Properties**.
5. Edit the free end cut properties as needed. For more information, see Free End Cut Properties Dialog Box.

Delete free end cut

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter**.
3. Select the free end cut to delete.
4. Click **Delete**.

Assembly Connection Properties Dialog Box

Specifies the properties for the member assembly connection that you are editing.

Occurrence Tab (Assembly Connection Properties Dialog Box) (on page 962)
Definition Tab (Assembly Connection Properties Dialog Box) (on page 966)
Relationship Tab (on page 123)
Configuration Tab (on page 124)
Notes Tab (on page 817)
Selection Tab (Assembly Connection Properties Dialog Box) (on page 966)

See Also

Edit assembly connection properties (on page 960)

Occurrence Tab (Assembly Connection Properties Dialog Box)

The **Occurrence** tab displays the member assembly connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection, and then selected the properties command, only the common properties between the selected assembly connections display.

When viewing properties for a single assembly connection, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on properties.

Name

Displays the name of the assembly connection. The assembly connection name is based on the **Name Rule** selection. To type a new name for the assembly connection, in the **Name Rule** box, select **User Defined**, and then type a name for the assembly connection in the **Name** box.

Name Rule

Specify the naming rule to use to name this assembly connection.

- **Default Name Rule** - Names the assembly connection using this format: <assembly connection>-<location>-<unique index> where <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, GussetPlateAsmConn_1-1-0045.
- **Unique Name Rule** - Names the assembly connection using this format: <parent system name>-<assembly connection>-<location>-<unique index> where <parent system name> is the parent System selected for the assembly connection, <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, Structure System-GussetPlateAsmConn_1-1-0045.
- **User Defined** - Select to specify the assembly connection name yourself in the **Name** box.

System

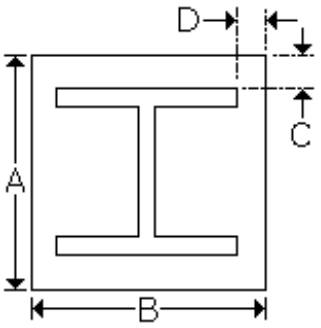
Select the system to which the assembly connection that you are placing belongs. You can

create new systems in the Systems and Specifications task.

Base Plate Assembly Connection Properties

Depth Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension C in the figure.



Width Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension D in the figure.

Sizing Rule

Select the sizing rule method for the base plate.

Plate Category

Select the plate category.

Plate Type

Select the plate type.

Miter Assembly Connection Properties

Top Distance

Specifies the distance between the top flange of the member section and the top of the plate.

Bottom Distance

Specifies the distance between the bottom flange of the member section and the bottom of the plate.

Left Distance

Specifies the distance between the left edge of the member section and the left edge of the plate.

Right Distance

Specifies the distance between the right edge of the member section and the right edge of the plate.

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

With Plates

Specifies whether or not a plate is inserted between the member ends. Select **False** to not place the plate. Select **True** to place the plate.

Clearance

Specifies the distance between the member ends.

Sizing Rule

Select the sizing rule method for the base plate.

Slab by Member Boundary

Clearance

Type a clearance distance between the edge of the slab and the member.

Port Face Position

Select the location on the member at which the slab is to stop. You can select the outmost plane, the centerline, or the in-most plane on the member.

Detailed Connection

Select **True** to trim the slab using the boundary member as the cutting edge. Select **False** to not trim the slab.

Offset

Specify the distance between the selected **Port Face Position** and the edge of the slab. A negative value moves the edge into the body of the slab. A positive value moves the edge out from the body of the slab.

Slab by Slab Assembly

Trim Type

Defines the type of trimming.

- **By Whole Slab** - Trim by the slab boundary geometry before trim.
- **By Lateral Face** - Trim by the slab boundary geometry after trim.
- **By Extended Lateral Face** - Trim by the extension of the lateral face of the slab boundary.

Trim Excess Material

Specify whether to trim the exceeding part.

Trim Excess Material Offset

Specify the offset applied on the lateral face used to trim.

Slab Free Edge Assembly

Reference Direction

Select the reference direction for the angle:

Normal

The angle is measured from a vector perpendicular to the slab edge.

- **Horizontal** - The angle is measured from the global XY plane in the model.
- **Vertical** - The angle is measured from the Z-Axis in the model.

Angle

Type a slope for the slab edge represented by the assembly connection. If the slab was place using the **Face Position Top**, the side face rotates about the top slab edge. If the slab was

placed using the **Face Position Bottom**, the side face rotates about the bottom slab edge.

Offset

Specify the distance between the selected boundary object and the edge of the slab.

Splice Assembly Connection Properties

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

Clearance

Specifies the distance between the ends of the members.

Splice With

Select the plates that you want to use in the splice.

Web Plate Position

Select a web plate position.

Distance from flange gage line

Specifies the distance from the flange gage line.

Distance from web gage line

Specifies the distance from the web gage line.

Flange Plate Thickness

Specifies the thickness of the flange plates.

Flange Plate Length

Specifies the length of the flange plates.

Flange Plate Width

Specifies the width of the flange plates.

Flange Plate Category

Select the plate category for the flange plates.

Flange Plate Type

Select the plate type for the flange plates.

Web Plate Thickness

Specifies the thickness of the web plates

Web Plate Length

Specifies the length of the web plates.

Web Plate Width

Specifies the width of the web plates.

Web Plate Category

Select the plate category for the web plates.

Web Plate Type

Select the plate type for the web plates.

General Surface Assembly Connection Properties

With Pad

Select **True** to place a steel pad between the member end and the surface. Select **False** to have the member end connect directly to the surface.

Pad Type

Select the shape of the pad.

Sizing Rule

Select whether you want to the software to automatically size the pad or if you want to size the pad.

Offset

Type the distance between the end of the member and the face of the surface.

See Also

Assembly Connection Properties Dialog Box (on page 962)

Definition Tab (Assembly Connection Properties Dialog Box)

The **Definition** tab displays the member assembly connection properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection and then selected the properties command, only the common properties between the selected assembly connections display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

See Also

Assembly Connection Properties Dialog Box (on page 962)

Selection Tab (Assembly Connection Properties Dialog Box)

Specifies the rule criteria used for the member assembly connection. Assembly connection rule selections affect the selections available for other smart occurrence objects that are children of the assembly connection.

TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.
- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.
- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.



1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.
2. Click the **Rule Based** box again to select and set all objects to rule based.

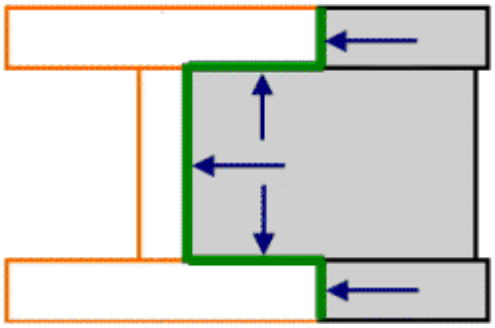
IMPORTANT

- The available questions, answers, and results vary depending upon the detailed parts associated with the connection.
- The questions in each **Selection** tab represent the default rules delivered with the software. Customized rules may have different questions.

SECTION 14

Place Member Generic Assembly Connection

Places an assembly connection between a member part and a bounding part. This command creates the assembly connection, applies the appropriate flange and web cuts, and creates the appropriate physical connections. A typical example is the bounding of a member to the flange and web faces of another member.



NOTE Member generic assembly connections are typically created automatically when a member is placed with **Place Linear Member Systems** and connected to a detailed plate system or edge reinforcement. You can use **Place Member Generic Assembly Connection** in situations where the connection is not created automatically.

The member is bounded to ports at the faces or edges of the bounding part. The ports used can be edited after creating the assembly connection. This command supports the following situations:

- A member bounded by a plate edge port.
- A member bounded by a plate face port.
- A member bounded by a flange edge port.
- A member bounded by a web or flange face port.

NOTES

- The selected bounding geometry must be on detailed parts.
- This command can generate physical connections to a single plate part.

Place Member Generic Assembly Connection Ribbon

Displays the options that you use to place an assembly connection between a built-up member part and intersecting face ports.

Properties

Activates the **Assembly Connection Properties** dialog box, which you use to view and modify the properties of the assembly connection. For more information, see *Assembly*

Connection Properties Dialog Box (on page 962).

Supported Member

Select the built-up member for which to create the assembly connection.

Supporting (Bounding) Geometry

Select a bounding port that intersects the end of the supported member to bound. If necessary, select additional bounding ports to which to create assembly and physical connections. Only those bounding ports that intersect the selected bounded end of the supported member are available for you to select.

Finish

Places the assembly connection.

Cancel

Clears the current selection.

Accept

Accepts the current selection. After you accept the supporting member and bounding geometry, the **Properties** button becomes active, and the **Assembly Connection** list is populated with the available connection types.

Assembly Connection

Select an assembly connection type from the list.

Rule Based

Enables or disables the **Assembly Connection** list. Selecting this check box disables **Assembly Connection** list.

Modify Member Generic Assembly Connection Ribbon

Displays the options that you use to modify an existing assembly connection between a built-up member part and intersecting face ports.

Properties

Activates the **Assembly Connection Properties** dialog box, which you use to view and modify the properties of the assembly connection. For more information, see *Assembly Connection Properties Dialog Box* (on page 962).

Supported Member

Displays the built-up member for which to create the assembly connection. The supported member cannot be modified.

Supporting (Bounding) Geometry

Select a bounding port that intersects the end of the supported member to bound. If necessary, select additional bounding ports to which to create assembly and physical connections. Only those bounding ports that intersect the selected bounded end of the supported member are available for you to select.

Finish

Places the assembly connection.

Cancel

Clears the current selection.

Accept

Accepts the current selection. After you accept the supporting member and bounding geometry, the **Properties** button becomes active, and the **Assembly Connection** list is populated with the available connection types.

Assembly Connection

Select an assembly connection type from the list.

Rule Based

Enables or disables the **Assembly Connection** list. Selecting this check box disables the **Assembly Connection** list.

What do you want to do?

- *Place a member generic assembly connection* (on page 970)
- *Modify a smart occurrence object* (on page 992)
- *Delete a smart occurrence object* (on page 992)
- *Edit assembly connection ports* (on page 971)

Place a member generic assembly connection

1. Click **Place Member Generic Assembly Connection**.
2. Select the member for which to create the assembly connection.
3. Select the bounding ports.

   TIPS

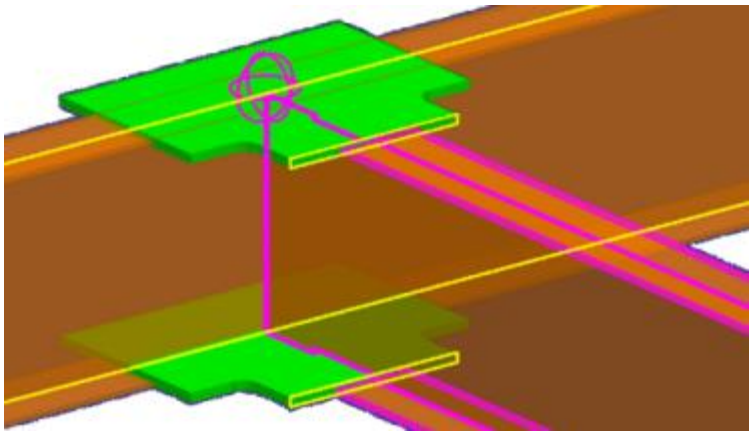
   - The software creates physical connections between the supported member and the bounding ports that you select.
   - Only those ports that intersect the selected end of the supported member are available for selection.
   - If you select only the top flange of the bounding built up, the software may not be able to completely trim the bounded member.
4. Click **Accept**.

   NOTE The **Rule Based** check box must be clear to select the assembly connection type.
5. Select the assembly connection type from the **Assembly Connection** list.
6. Click **Finish**.

Edit assembly connection ports

1. In a graphic view or in the **Workspace Explorer**, select a member with an assembly connection that you want to modify.

   The member and assembly connection highlight in magenta. The assembly connection ports highlight in yellow.



2. Select the port that you want to change.

   *The **Modify Member Generic Assembly Connection** ribbon appears with **Supporting (Bounding) Geometry** as the active option.*

3. To remove an existing port, select it.

   The port is no longer yellow.

4. To add a new port, select it.

   The port highlights in yellow.

5. Click **Accept**.

6. If you want to change the assembly connection type select a new value from the **Assembly Connection** list.

7. Click **Finish**.

NOTES

- A list of members and plates connected to the selected member can be viewed on the **Relationship** tab of the **Assembly Connection Properties** dialog box. Select a connected object from the list to highlight it in the graphic view.
- Because you can modify the bounding geometry ports, but not the bounded member, the **Supported Member** option is not available.

SECTION 15

Creating Reference Curves

The commands in the reference curves category allow you to create and modify 3D curves defined on any planar, knuckled, or curved surface. You can classify curves as reference, grids, marking, tangents, and knuckles.



NOTES

- When you create or edit a reference curve, the dialog box title displays the plate system name using the format **Plate System:** *<name of the reference plane>*.



- You can change the default colors of reference curves, seams, profile systems, and knuckle curves to enhance visibility. For more information, see *Modify Reference Curve Display Colors* (on page 986).



Reference Curve by Intersection - Creates a reference curve on the selected plate system by defining the landing curve as the intersection of the selected plate system and a grid plane. For more information, see *Reference Curve by Intersection* (on page 973).



Reference Curve by Offset - Creates a reference curve on the selected plate system by defining the landing curve as being offset from an existing curve (such as another seam, plate boundary, profile system, or grid plane). For more information, see *Reference Curve by Offset* (on page 976).



Reference Curve by 2D Sketch - Creates a reference curve on the selected plate system by projecting a 2D landing curve that you have sketched onto the plate system. For more information, see *Reference Curve by 2D Sketch* (on page 980).



Reference Curve from External File - Creates and modifies knuckled reference curves. For more information, see *Reference Curve from External File* (on page 984).

See Also

Knuckle Best Practices (on page 56)
Landing Curve Best Practices (on page 56)

Reference Curve by Intersection

The **Reference Curve by Intersection** command creates a reference curve on the selected plate system by defining the landing curve as the intersection of the selected plate system and a grid plane.

You can control the boundaries of the reference curve by selecting a plate system or reference curve that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the reference curve boundaries. The reference curve must end at a plate system boundary or at another reference curve on the same plate system. The reference curve can cross an opening in the plate system.

Reference Curve by Intersection Ribbon

Displays the options that you use to place a reference curve by intersection in the model.

Reference Curve by Intersection Properties

Opens the **Reference Curve by Intersection Properties** dialog box, which you use to view and modify the properties of the reference curve that you are about to place in the model. For more information, see *Reference Curve by Intersection Properties Dialog Box* (on page 975).

Plate System

Specifies the plate systems. You can select multiple plate systems to place multiple reference curves at the same time. The software uses the same boundaries for each reference curve. After placement, the reference curves are unrelated to each other and can be edited as individual entities.

Intersecting Objects

Specifies the objects that intersect the plate system for which you want to generate seams.

Boundaries

Defines the boundaries of the seam. You can define the boundaries by selecting objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the *Define Boundary Controls* section below.

Finish

Creates the seam using the defined parameters.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Type

Specifies the curve type. Select **Reference**, **Grid**, **Mark**, **Tangent**, or **Knuckle**.

Name

Defines the name of the reference curve.

Define Boundary Controls

Available when you use the **Boundaries** option.

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

What do you want to do?

- *Place a reference curve by intersection* (on page 974)
- *Modify a reference curve* (on page 975)
- *Delete a reference curve* (on page 975)

Place a reference curve by intersection

1. Click **Reference Curve by Intersection** on the vertical toolbar.
2. Select the plate system to receive the reference curve.
3. Click **Intersecting Objects**.
4. Select an intersecting grid plane or intersecting plate system where you want the reference curve.
5. Click **Boundaries**.
6. Define the reference curve boundaries using one or more of the following methods:

   Pick boundaries (on page 81)

   Define a boundary curve (on page 82)
7. Click **Accept**.
8. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

   Solve ambiguous solution created by selected boundaries (on page 82)
9. Click **Finish** to place the reference curve.

NOTES

- If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting reference curve.

- A reference curve is always attached to a plate system.

Modify a reference curve

1. Click **Select** on the vertical toolbar.
2. Select **RefCurve on Surface & Knuckles** in the **Locate Filter**.
3. Select the reference curve to modify.
4. Edit the reference curve as needed using the ribbon bar.

Delete a reference curve

1. Click **Select** on the vertical toolbar.
2. Select **RefCurve On Surface & Knuckles** in the **Locate Filter** box.
3. Select the reference curve to delete.
4. Click **Edit > Delete**.

Reference Curve By Intersection Properties Dialog Box

Specifies the properties for the reference curve that you are placing or editing.

Main Tab (Reference Curve By Intersection Properties Dialog Box) (on page 975)
Configuration Tab (on page 124)
General Tab (on page 125)

Main Tab (Reference Curve By Intersection Properties Dialog Box)

Specifies the general properties of the reference curve.

Name

Specifies the name of the reference curve.

Rule

Select the naming rule used to name the reference curve. Select from the following:

- **StdHierarchyChildNamingRule** - This name is generated by using the name of the Parent system as a prefix, followed by the interface name and a unique number, generated by the system and appended to the name; for example, *Hull-IJRefCurveOnSurface101.*
- **StdReferenceCurveNamingRule** - The name is generated according to the curve type:
  - **Grid Curve** - Uses the names of the Coordinate System and Grid Plane as a prefix; for example, *Grid:CS_Main:F16-201*.
  - **Mark Curve** - Uses the prefix *Mark*; for example, *Mark-110*.
  - **Reference Curve** - Uses the prefix *Reference;* for example, *Reference-110.*
  - **Tangent Curve** - Uses the prefix *Tangent;* for example, *Tangent-110.*

- **Knuckle Curve** - Uses the prefix *Knuckle;* for example, *Knuckle-110.*

- **User Defined** - Any text can be assigned; for example, *Hull1*.

Type

Specifies the type of marking information to be placed on parts.

- **Reference** - Use for general purpose.
- **Grid** - (grid plane curve). A reference curve created using **Intersection with Grid Plane** is automatically created as the curve type of grid. However, you could change the type. Use to mark off sections of a hull.
- **Mark** - (marking curve). Use in the manufacturing process for special markings such as "No Smoking."
- **Tangent** - (tangent curve). Use in the manufacturing process for special markings for processing and unfolding (such as roll boundaries).
- **Knuckle** - (knuckle curve). Use to control how the plate system is handled in Molded Forms regarding bending or splitting along the knuckle curve.

Landing Curve Definition Method

Displays the landing curve definition.

Parent System

Displays the name of the parent system.

Reference Curve by Offset

The **Reference Curve by Offset** command creates a reference curve on the selected plate system by defining the landing curve as being offset from an existing curve (such as another seam, plate boundary, profile system, or grid plane).

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the reference curve boundaries. Reference curves must end at a plate system boundary or at another reference curve on the same plate system.

NOTE The reference curve can cross an opening in the plate system.

Reference Curve by Offset Ribbon

Displays the options that you use to place an offset reference curve in the model.

Reference Curve by Offset Properties

Opens the **Reference Curve by Offset Properties** dialog box, which you use to view and modify the properties of the reference curve that you are about to place in the model. For more information, see *Reference Curve by Offset Properties Dialog Box* (on page 979).

Plate System

Specifies the plate system in which you want to place a reference curve. You can select multiple plate systems to place multiple reference curves at the same time. The software uses the same information for each reference curve. After placement, the reference curves are unrelated to each other and can be edited as individual entities.

Base Curve

Specifies an existing curve or grid plane from which to offset the reference curve. The intersection of the grid plane and the plate system defines the base curve.

Offset

Defines the offset parameters and direction from the base curve. See the *Offset Curve Definition Controls* section below.

Boundaries

Defines the boundaries of the reference curve. You can define the boundaries by selecting objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the reference curve to maintain the boundary relationship. See the *Boundary Definition Controls* section below.

Finish

Places the reference curve using the defined parameters.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Type

Specifies the curve type. Select **Reference**, **Grid**, **Mark**, **Tangent**, or **Knuckle**.

Name

Defines the name of the reference curve.

Offset Curve Definition Controls

Unlimited

Defines offset curves without a stop limit.

Stop Limit

Defines offset curves with a stop limit. By using a stop limit, you can select an edge, profile landing curve, connection reference curve, or reference curve to control the landing curves being created. If you type an offset value and the **Offset lock** is locked, then landing curves are applied at increments of the offset value until the selected stop limit is reached. If the **Offset lock** is unlocked and a value has been defined for **Count**, then the landing curves equal to the number entered in the **Count** box are equally spaced between the base curve and the stop limit.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by mouse moves.

Offset

Specifies the offset distance between the base curve and the landing curve that you are defining. If you type **0**, the landing curve is place on the base curve. This box may also display the offset distance as defined by mouse movements.

Method

Specifies an offset method. You can select transverse, longitudinal, vertical, or girth.

Count

Specifies how many reference curves are created, each equally spaced by increments of the offset value.

Step

Defines an increment that the software uses as you dynamically move the mouse. If you entered a value in the **Offset** box, then this control is not used.

Boundary Definition Controls

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

What do you want to do?

- *Place a reference curve by offset* (on page 978)
- *Modify a reference curve* (on page 975)
- *Delete a reference curve* (on page 975)

Place a reference curve by offset

1. Click **Reference Curve by Offset** on the vertical toolbar.
2. Select the plate system.
3. Click **Base Curve**.
4. Select a base curve that the software is to use to compute the offset curve.
5. Click **Offset**.
6. Type an offset value in the **Offset** box.
7. Specify the offset method and count to use.
8. Select, with a mouse click in a graphic view, on which side of the base curve to apply the offset.

   A red arrow appears as you move the mouse indicating the direction of the offset.

9. Click **Boundaries** .
10. Define the seam boundaries using one or more of the following methods:

    Pick boundaries (on page 81)
    Define a boundary curve (on page 82)
11. Click **Accept** .
12. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)
13. Click **Finish** to place the seam.

NOTES

- If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting reference curves.
- A reference curve is always attached to a plate system.

Reference Curve By Offset Properties Dialog Box

Specifies the properties for the reference curve that you are placing or editing.

Main Tab (Reference Curve By Offset Properties Dialog Box) (on page 979)
Configuration Tab (on page 124)
General Tab (on page 125)

Main Tab (Reference Curve By Offset Properties Dialog Box)

Specifies the general properties of the reference curve.

Name

Specifies the name of the reference curve.

Rule

Select the naming rule used to name the reference curve. Select from the following:

- **StdHierarchyChildNamingRule** - This name is generated by using the name of the Parent system as a prefix, followed by the interface name and a unique number, generated by the system and appended to the name; for example, *Hull-IJRefCurveOnSurface101.*
- **StdReferenceCurveNamingRule** - The name is generated according to the curve type:
  - **Grid Curve** - Uses the names of the Coordinate System and Grid Plane as a prefix; for example, *Grid:CS_Main:F16-201*.
  - **Mark Curve** - Uses the prefix *Mark*; for example, *Mark-110*.
  - **Reference Curve** - Uses the prefix *Reference;* for example, *Reference-110.*
  - **Tangent Curve** - Uses the prefix *Tangent;* for example, *Tangent-110.*
  - **Knuckle Curve** - Uses the prefix *Knuckle;* for example, *Knuckle-110.*
- **User Defined** - Any text can be assigned; for example, *Hull1*.

Type

Specifies the type of marking information to be placed on parts.

- **Reference** - Use for general purpose.
- **Grid** - (grid plane curve). A reference curve created using **Intersection with Grid Plane** is automatically created as the curve type of grid. However, you could change the type. Use to mark off sections of a hull.
- **Mark** - (marking curve). Use in the manufacturing process for special markings such as "No Smoking."
- **Tangent** - (tangent curve). Use in the manufacturing process for special markings for processing and unfolding (such as roll boundaries).
- **Knuckle** - (knuckle curve). Use to control how the plate system is handled in Molded Forms regarding bending or splitting along the knuckle curve.

Landing Curve Definition Method

Displays the landing curve definition.

Parent System

Displays the name of the parent system.

Reference Curve by 2D Sketch

 The **Reference Curve by 2D Sketch** command creates a reference curve on the selected plate system by projecting a 2D landing curve that you have sketched onto the plate system.

You can control the boundaries of the reference curve by selecting a plate system, or reference curve that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the reference curve boundaries. Reference curves must end at a plate system boundary or at another reference curve on the same plate system. The reference curve can cross an opening in the plate system.

Reference Curve by 2D Sketch Ribbon

Displays the options that you use to place a reference curve in the model.

 **Reference Curve by 2D Sketch Properties**

Opens the **Reference Curve by 2D Sketch** dialog box, which you use to view and modify the properties of the reference curve that you are about to place in the model.

 **Plate System**

Select the plate system for the reference curve. You can select multiple plate systems to place multiple reference curves at the same time. The software uses the same information for each reference curve. After placement, the reference curves are unrelated to each other and can be edited as individual entities.

 **Sketch Curve**

Select to draw the landing curve. See the *Landing Curve Definition Controls* section below.

 **Boundaries**

Define the boundaries of the reference curve. You can define the boundaries by selecting

objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the reference curve to maintain the boundary relationship. See the *Boundary Definition Controls* section below.

Finish

Places the reference curve using the defined parameters.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Type

Specifies the curve type. Select **Reference**, **Grid**, **Mark**, **Tangent**, or **Knuckle**.

Name

Defines the name of the reference curve.

Landing Curve Definition Controls

Available when you use the **Sketch Curve** option.

Sketching Plane

Specify the sketching plane for the curve. This is the first step in defining the curve. See the *Define Plane Controls* section below.

Add Intersecting Item

Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the Sketch 2D environment.

Add Projection Item

Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the curve.

NOTE To display a grid or reference plane ruler in the Sketch 2D environment, first define an active coordinate system using **Tools** > **PinPoint**. For more information, see PinPoint in the *Common User's Guide* and *2D Grids Ruler* in the *SmartSketch Drawing Editor Help*.

Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Project Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Boundary Definition Controls

Available when you use the **Boundaries** option.

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 83).

What do you want to do?

- *Place a reference curve by 2D sketch* (on page 982)
- *Modify a reference curve* (on page 975)
- *Delete a reference curve* (on page 975)

Place a reference curve by 2D sketch

1. Click **Reference Curve by 2D Sketch** on the vertical toolbar.
2. Select the plate system for the reference curve.
3. Click **Sketch Curve**.
4. Specify one plane for the reference curve sketch using one of the following methods:

   Define a coincident plane (on page 76)
   Define an offset plane (on page 77)
   Define plane using angle from plane
   Define a plane using a point and a normal vector (on page 78)
   Define a plane using three points (on page 79)
   Define plane by vectors normal
5. Click **Add Intersecting Item**.
6. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the 2D environment.
7. Click **Add Projection Item**.
8. Select objects in the model that do not intersect the sketching plane that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the 2D environment.
9. Click **Sketch 2D**.

10. In the 2D environment, sketch the reference curve.

    NOTE You can constrain the curve to geometric construction points offset from plate systems. For more information, see *Point at Minimum Distance Command* in the *SmartSketch Drawing Editor Drawing Editor Help*, available within the 2D environment.

11. Click **Finish** in the 2D environment.
12. Click **Boundaries** .
13. Define the reference curve boundaries using one or more of the following methods:

    Pick boundaries (on page 81)
    Define a boundary curve (on page 82)

14. Click **Accept** .
15. If the boundary definition results in two solutions, pick the needed solution using the following procedure:

    Solve ambiguous solution created by selected boundaries (on page 82)

16. Click **Finish** to place the reference curve.

NOTES

- If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting reference curves.
- A reference curve is always attached to a plate system.

Reference Curve By 2D Sketch Properties Dialog Box

Specifies the properties for the reference curve that you are placing or editing.

Main Tab (Reference Curve By 2D Sketch Properties Dialog Box) (on page 983)
Configuration Tab (on page 124)
General Tab (on page 125)

Main Tab (Reference Curve By 2D Sketch Properties Dialog Box)

Specifies the general properties of the reference curve.

Name

Specifies the name of the reference curve.

Rule

Select the naming rule used to name the reference curve. Select from the following:

- **StdHierarchyChildNamingRule** - This name is generated by using the name of the Parent system as a prefix, followed by the interface name and a unique number, generated by the system and appended to the name; for example, *Hull-IJRefCurveOnSurface101.*
- **StdReferenceCurveNamingRule** - The name is generated according to the curve type:
  - **Grid Curve** - Uses the names of the Coordinate System and Grid Plane as a prefix; for example, *Grid:CS_Main:F16-201*.
  - **Mark Curve** - Uses the prefix *Mark*; for example, *Mark-110*.

- **Reference Curve** - Uses the prefix *Reference;* for example, *Reference-110.*
- **Tangent Curve** - Uses the prefix *Tangent;* for example, *Tangent-110.*
- **Knuckle Curve** - Uses the prefix *Knuckle;* for example, *Knuckle-110.*

- **User Defined** - Any text can be assigned; for example, *Hull1*.

Type

Specifies the type of marking information to be placed on parts.

- **Reference** - Use for general purpose.
- **Grid** - (grid plane curve). A reference curve created using **Intersection with Grid Plane** is automatically created as the curve type of grid. However, you could change the type. Use to mark off sections of a hull.
- **Mark** - (marking curve). Use in the manufacturing process for special markings such as "No Smoking."
- **Tangent** - (tangent curve). Use in the manufacturing process for special markings for processing and unfolding (such as roll boundaries).
- **Knuckle** - (knuckle curve). Use to control how the plate system is handled in Molded Forms regarding bending or splitting along the knuckle curve.

Landing Curve Definition Method

Displays the landing curve definition.

Parent System

Displays the name of the parent system.

Reference Curve from External File

Reference Curve from External File creates and modifies knuckle reference curves. These curves require that the edges of the sheetbody faces carry an attribute that reflects that they are knuckles and relates them to the appropriate reference curve.

Reference Curve from External File Ribbon

Reference Curve Properties

Opens the **Reference Curve by Importing Properties** dialog box, which you use to view and modify the properties of the reference curve that you are about to place in the model. For more information, see *Reference Curve from External File Properties Dialog Box* (on page 985).

Parent Plate System

Specifies the plate system onto which you want to place the reference curve.

Open external file

Specifies the SAT or IGS file containing the reference curve to place.

Finish

Places the reference curve using the defined parameters.

Point Tolerance

Specifies the distance that is used to determine whether the surface edges are aligned with the reference curve.

Import Options

Modify existing Reference Curves

When selected, modifies existing curves during the import.

Delete missing Reference Curves

When selected, deletes missing curves during the import.

What do you want to do?

- *Place a reference curve by importing from file* (on page 985)
- *Modify a reference curve* (on page 975)
- *Delete a reference curve* (on page 975)

Place a reference curve by importing from file

1. Click **Reference Curve from External File** on the vertical toolbar.
2. Select the parent plate system.
3. Type a value for **Point Tolerance**.
4. Click **Open external file**, and select the file to import.
5. Click **Finish**.

NOTES

- If you do not select boundaries, the software applies the entire length of the base curve to the resulting reference curves.
- A reference curve is always attached to a plate system.

Reference Curve from External File Properties Dialog Box

Specifies the properties for the reference curve that you are placing or editing.

Main Tab (Reference Curve from External File Properties Dialog Box) (on page 985)
Configuration Tab (on page 124)
General Tab (on page 125)

Main Tab (Reference Curve from External File Properties Dialog Box)

Specifies the general properties of the reference curve.

Name

Specifies the name of the reference curve.

Rule

Select the naming rule used to name the reference curve. Select from the following:

- **StdHierarchyChildNamingRule** - This name is generated by using the name of the Parent system as a prefix, followed by the interface name and a unique number, generated by the system and appended to the name; for example, *Hull-IJRefCurveOnSurface101.*
- **StdReferenceCurveNamingRule** - The name is generated according to the curve type:
  - **Grid Curve** - Uses the names of the Coordinate System and Grid Plane as a prefix; for example, *Grid:CS_Main:F16-201*.
  - **Mark Curve** - Uses the prefix *Mark*; for example, *Mark-110*.
  - **Reference Curve** - Uses the prefix *Reference;* for example, *Reference-110.*
  - **Tangent Curve** - Uses the prefix *Tangent;* for example, *Tangent-110.*
  - **Knuckle Curve** - Uses the prefix *Knuckle;* for example, *Knuckle-110.*
- **User Defined** - Any text can be assigned; for example, *Hull1*.

Type

Specifies the type of marking information to be placed on parts.

- **Reference** - Use for general purpose.
- **Grid** - (grid plane curve). A reference curve created using **Intersection with Grid Plane** is automatically created as the curve type of grid. However, you could change the type. Use to mark off sections of a hull.
- **Mark** - (marking curve). Use in the manufacturing process for special markings such as "No Smoking."
- **Tangent** - (tangent curve). Use in the manufacturing process for special markings for processing and unfolding (such as roll boundaries).
- **Knuckle** - (knuckle curve). Use to control how the plate system is handled in Molded Forms regarding bending or splitting along the knuckle curve.

Landing Curve Definition Method

Displays the landing curve definition.

Parent System

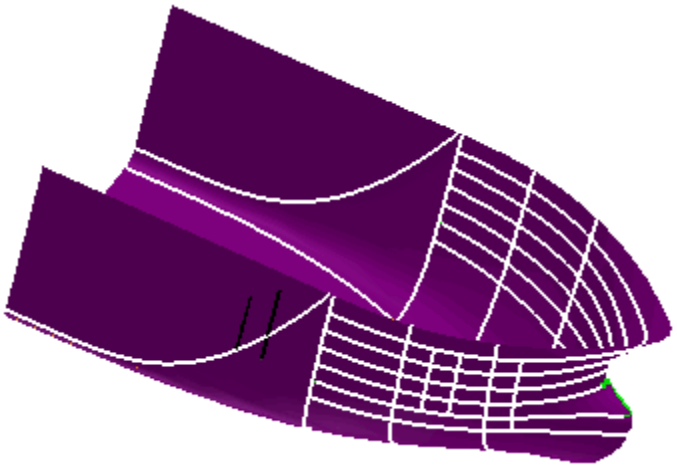
Displays the name of the parent system.

Modify Reference Curve Display Colors

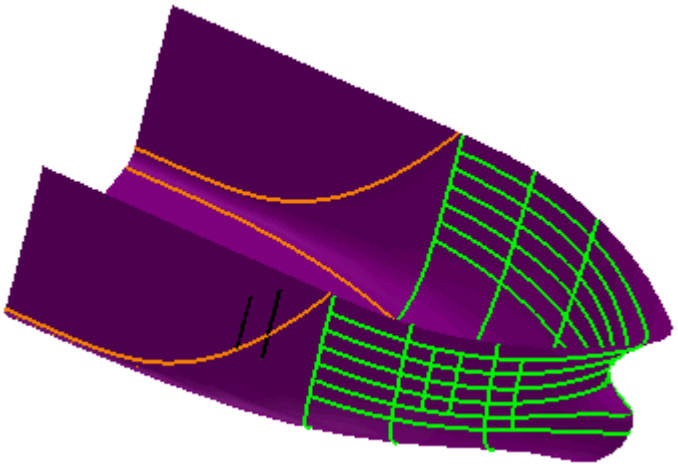
Use this procedure to create style rules to customize the display colors of landing curves. You can use the procedure to set different colors for the landing curves of:

- Seams
- Profile Systems
- Reference Curves
- Plate Knuckle Curves

For example, before creating style rules, reference curves and design seams are white:



After creating separate style rules for reference curves and design seams, they appear in different colors:



Use the following steps to customize curve display colors.

Select the View Aspect

1. Click **Format** >**View**.

   *The **Format View** dialog box opens.*

2. In the **Selected Aspects** box, click **Reference Geometry**.

   NOTE By default, **Simple physical** is also selected.

3. Click **OK**.

Create a Style Rule

1. Click **Format** > **Surface Style Rules**.

   *The **Surface Style Rules** dialog box opens.*

2. Click **New**.

   *The **Surface Style Rule Properties** dialog box opens*

3. Type a name for the new surface style rule in the **Rule name** box.
4. In the **Filter** list, click **More** to create a new filter.

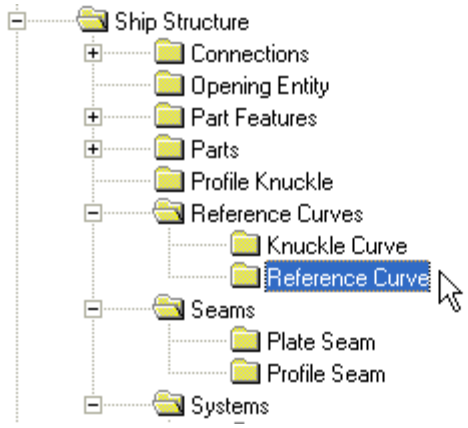
   *The **Select Filter** dialog box opens.*

5. Click **New Filter**.

   *The **New Filter Properties** dialog box opens.*

   NOTE If you are an administrator, you can first select **Ship Filters**. The new filter is then available to all users.

6. On the **Object Type** tab, expand the hierarchy **Ship Structure**, and select one of the following:
   - Ship Structure/ Reference Curves/**Reference Curve**
   - Ship Structure/ Reference Curves/**Knuckle Curve**
   - Ship Structure/Seams/**Plate Seam**
   - Ship Structure/Systems/Profile Systems/**Stiffener System**



7. Click **OK**.
8. Click **OK** in the **Select Filter** dialog box.
9. In the **Surface Style Rule Properties** dialog box, select a color in the **Style applied** box.
10. Click **OK**.

    *The new rule appears in the **Style rule library** list.*

Add the Rule to the Workspace

1. Select the rule in the **Style rule library** list.
2. Click **Add**.
3. Click **OK**.

SECTION 16

Execute Detailing Command

 The **Execute Detailing** command creates detail parts from the root part of a plate, profile, or member system. The command requires you to select one or more blocks, assemblies, systems, sub-systems, or root parts. You can select built-up members for detailing, but you cannot select rolled members. You can select the objects from the model, or from the **Workspace Explorer**.

Objects can be selected for detailing both before and after launching the **Execute Detailing** command. Only valid items are added to the list and are highlighted in yellow, and invalid commands are filtered out.

After you run the **Execute Detailing** command, the icon for the object changes in the **Workspace Explorer**. For more information, see *Icons in the Workspace Explorer* (on page 32).

When parts are created in the Molded Forms task, those parts are trimmed to the molded surfaces of the part's boundaries. Parts in this initial state are referred to as *light* parts. The **Execute Detailing** command transforms the light parts to production definition and accuracy by considering the thickness, orientation, and connection information of the bounding objects. Parts in this advanced state are referred to as *detailed* parts. In addition to the thickness based trimming, other objects, such as assembly and physical connections, slots, collars, end cuts, and chamfers are automatically created. These additional objects are either incorporated in the detail part, or become related to the detailed part, as appropriate.

The software trims a detail part based on its bounding objects. After a bounding object has been detailed, the resulting detail part boundary is trimmed to the face of the bounding detail part, which means the boundary is adjusted by the thickness of the bounding detail part. If the bounding object has not been detailed (that is, the **Execute Detailing** command has not been run for its root part), then the resulting detail part boundary is trimmed to the bounding parent plate system surface. When you run **Execute Detailing** against the bounding root part, the boundary of the bounded detail part is adjusted as required by the molded thickness direction and the thickness of the bounding detail part.

As parts are detailed, additional detailing objects are created as smart occurrences by rules in the reference data and can be individually modified after creation. Assembly and physical connections are created for both parts associated with a connection. Free end cuts are created at the ends of unbounded profiles or members. If you have run **Execute Split** in the Molded Forms task, the software automatically creates slot and collar features where stiffeners penetrate plates, end cuts on profiles, and chamfers at design seams where plate thickness change by a specified amount. The default values of smart occurrence objects cannot be modified with the **Execute Detailing** command, but can be modified after creation by selecting the object and modifying the properties of the object.

If you select a detailed part and click **Delete**, the software removes the detailing from that part.

If you have run **Execute Detailing** for only one of the root parts, then the software does not create the assembly and physical connections or any applicable features. For example, if you run **Execute Detailing** for a root part that is penetrated by a stiffener that has not had **Execute Detailing** run, then the assembly and physical connections for the penetration are not created, and thus the slot and collar or clip features created by the assembly connection are also not created. When you run **Execute Detailing** for the penetrating root part, the assembly connection at the penetration is created. The software then creates the slot on the penetrated detail part.

The generated detail parts inherit their properties from the parent plate or profile system.

NOTE Straking seams can only be placed on detail parts. Therefore, you must run **Execute Detailing** before you can place straking seams.

Execute Detailing Ribbon

Displays the controls used to generate detail parts.

Select objects for detailing

Select the objects from which to generate detail parts. You can select the objects in a graphic view or in the **Workspace Explorer**.

Finish

Creates the detail parts using the parameters that you have defined.

Plate Systems and Parts Only

Excludes profiles from the parts that the software processes. If you select this option before you select parts, the software prevents you from selecting profile objects. If you select this option before you click **Finish**, the software excludes any profiles that are in your selected objects from processing.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Propagate changes

In addition to detailing the selected objects, other parts affected by the detailing are also changed. If many parts are affected, propagation can significantly impact processing time.

Do not propagate changes

Only the selected objects are detailed. Other parts affected by the detailing are not changed, keeping processing time to a minimum.

NOTES

- You can select plate or profile parts by default in a graphic view, or from the **Workspace Explorer**.
- You can select planning blocks by default in a graphic view, or from the **Workspace Explorer**.
- You can select plate or profile systems and sub systems in the **Workspace Explorer** or by using **QuickPick** in a graphic view. **Execute Detailing** details all child parts.
- You can select planning blocks and assemblies in the **Workspace Explorer** or by using **QuickPick** in a graphic view. **Execute Detailing** details all assigned parts.
- Select the **Plate Systems & Parts Only** option to exclude profiles from the objects that you can select.
- If you select root parts, then only the root parts selected are detailed. For example, selecting a root plate part does not detail the root profile parts on the plate, even if **Plate Systems and Parts Only** is turned off.

Smart Occurrence Ribbon

Displays the controls used to modify a smart occurrence object. The ribbon is available after the smart occurrence object is created automatically by the **Execute Detailing** command.

Properties

Activates the **Properties** dialog box, which you use to view and modify the properties of the smart occurrence object that you are modifying. The initial properties default from the Structural Detailing rules used by the **Execute Detailing** command.

Item

Displays the final result of User Answers on the **Selection** tab of the **Properties** dialog box.

NOTE **Item** does not display for some smart occurrence objects.

Modify Auxiliary Parts Ribbon

Displays the options that you use to add auxiliary parts to the selected assembly connection.

Properties

Activates the **Assembly Connection Properties Dialog Box**, which you use to view and modify the properties of the assembly connection before the assembly connection is committed to the database. For more information, see *Assembly Connection Properties Dialog Box* (on page 962).

Assembly Connection Smart Item

Displays valid smart items for the selected assembly connection.

Add/Delete Auxiliary Parts

Select the parts that can be connected to the selected assembly connection and do not already have an existing assembly connection. Parts that do not have common geometry with the selected assembly connection cannot be selected.

Accept

Adds parts from selected parts to the assembly connection and saves the changes to the database.

Reject

Removes any selected parts from the auxiliary parts list. Only selections made after the last commit are removed.

What do you want to do?

- *Assign part names at placement* (on page 992)
- *Create detailed parts from plate and profile systems* (on page 992)
- *Delete a smart occurrence object* (on page 992)
- *Modify a smart occurrence object* (on page 992)
- *Remove detailed parts from plate and profile systems* (on page 993)
- *Add auxiliary parts to an assembly connection* (on page 960)

- *Edit assembly connection parts* (on page 961)

Assign part names at placement

Every task that creates new parts in the model must assign a part name to each new part.

In the Structural Detailing task, the part name is automatically generated using the default name rule when you place the part. The name consists of the system parent, the category of the part, and a sequencing number.

You can change the part name on the Main Tab for each part. When you override the automatically generated name in the **Name** box on the **Main** tab, the text in the **Rule** box changes to **User Defined**.

Create detailed parts from plate and profile systems

1. Click **Execute Detailing** .
2. Select the parts to detail.

   TIPS

   - You can select plate or profile parts by default in a graphic view, or from the **Workspace Explorer**.
   - You can select planning blocks by default in a graphic view, or from the **Workspace Explorer**.
   - You can select plate or profile systems and sub systems in the **Workspace Explorer** or by using **QuickPick** in a graphic view. **Execute Detailing** details all child parts.
   - You can select planning blocks and assemblies in the **Workspace Explorer** or by using **QuickPick** in a graphic view. **Execute Detailing** details all assigned parts.
   - Select the **Plate Systems & Parts Only** option to exclude profiles from the objects that you can select.
   - If you select root parts, then only the root parts selected are detailed. For example, selecting a root plate part does not detail the root profile parts on the plate, even if **Plate Systems and Parts Only** is turned off.
3. Click **Finish**.

Delete a smart occurrence object

1. Click **Select** on the vertical toolbar.
2. Select the object to delete.
3. Click **Delete** .

Modify a smart occurrence object

1. Click **Select** on the vertical toolbar.

2. Select one of the following in the **Locate Filter** box:
   - **Features** to select slots, collars, web cuts, flange cuts, or chamfers
   - **Assembly Connections / Free End Cuts**
   - **Physical Connections**
3. Select the smart occurrence object to modify.
4. Click **Properties** .
5. Edit properties and parameters for the object as needed.

Remove detailed parts from plate and profile systems

1. Select one or more detailed parts.

   NOTE Detailed parts have red icons in the **Workspace Explorer**. For more information, see *Icons in the Workspace Explorer* (on page 32).
2. Click **Delete** .

   The detailed part is deleted, and replaced by a light part.

Detailing Built Up Connections

The following cases provide examples of the results that you can expect when the software details member to member connections involving built up (designed) members.

NOTE The following abbreviations are used in these descriptions:

- M# - Member System
- FC# - Frame Connection
- DM# - Designed Member
- P# - Plate System
- AC# Assembly Connection
- PC# Physical Connection
- P#' - Plate sub System
- PP# - Plate Part
- LC# Logical Connection
- LC#' - Sub Logical Connection

Topics

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Tube End to Tube End - Co-Linear Axis 998
Tube to Tube - Axis Along 1000
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Designed Member to Designed Member - Axis Along 1003
Designed Member to Designed Member - Axis Along 1004
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Can - Continuous Leg.....1021
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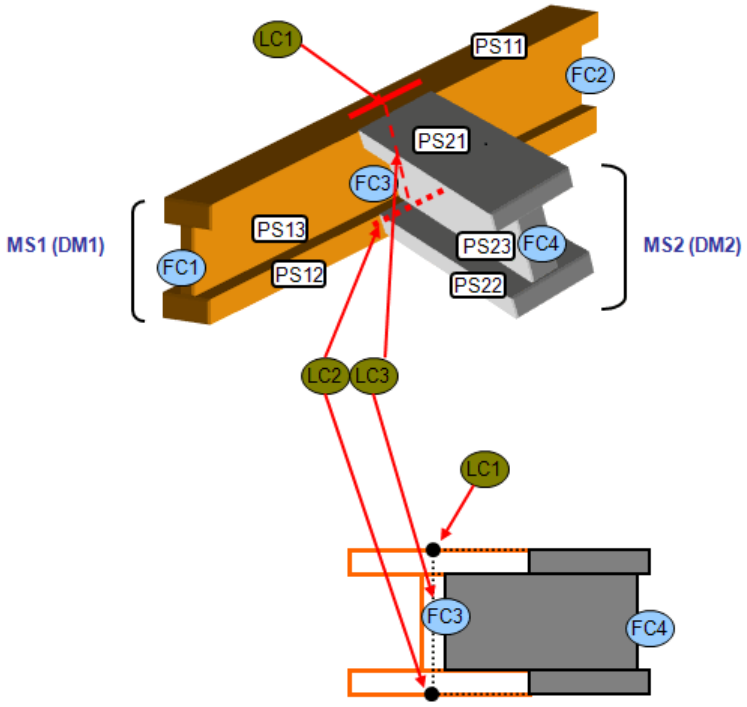
Designed Member to Designed Member - Axis Along

Member2 (DM2) to Member1 (DM1) Axis Along

Object Hierarchy Before Detailing

```
MS1
  -DM1
    -PS11
      -PS11'
        -PP11
    -PS12
      -PS12'
        -PP12
    -PS13
      -PS13'
        -PP13
  -FC1
```

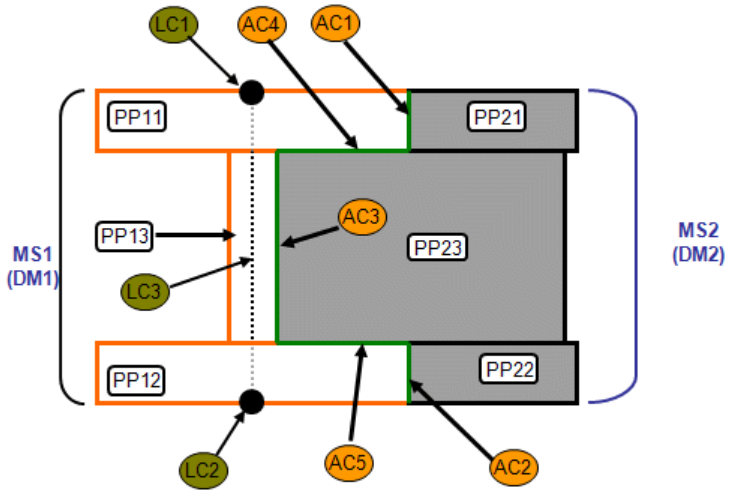
```
  -FC2
MS2
  - DM2
    - PS21
      - PS21'
        - PP21
      - LC 1 (PS21 <-> MS1)
        - LC' 1 (PS21'<-> MS1)
    - PS22
      - PS22'
        - PP22
      - LC 2 (PS22 <-> MS1)
        - LC' 2 (PS22'<-> MS1)
    - PS23
      - PS23'
        - PP23
      - LC 3 (PS23 <-> MS1)
        - LC' 3 (PS23'<-> MS1)
  - FC3
  - FC4
```



Object Hierarchy After Detailing

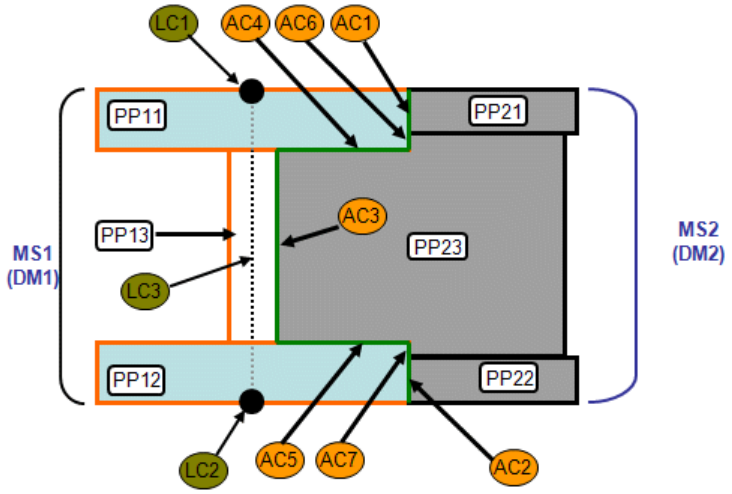
```
MS1
  - …
MS2
  - DM2
    - PS21
      - PS21'
        - PP21
```

```
    - LC 1
      - LC' 1
        - AC1 (PP21 <-> PP11)
          - PC1
  - PS22
    - PS22'
      - PP22
    - LC 2
      - LC' 2
        - AC2 (PP22 <-> PP12)
          - PC2
  - PS23
    - PS23'
      - PP23
    - LC 3
      - LC' 3
        - AC3 (PP23 <-> PP13)
          - PC3
        - AC4 (PP23 <-> PP11)
          - PC4
        - AC5 (PP23 <-> PP12)
          - PC5
- FC3
- FC4
```



Flange Thickness of Supported Member is Less Than Flange Thickness of Supporting Member

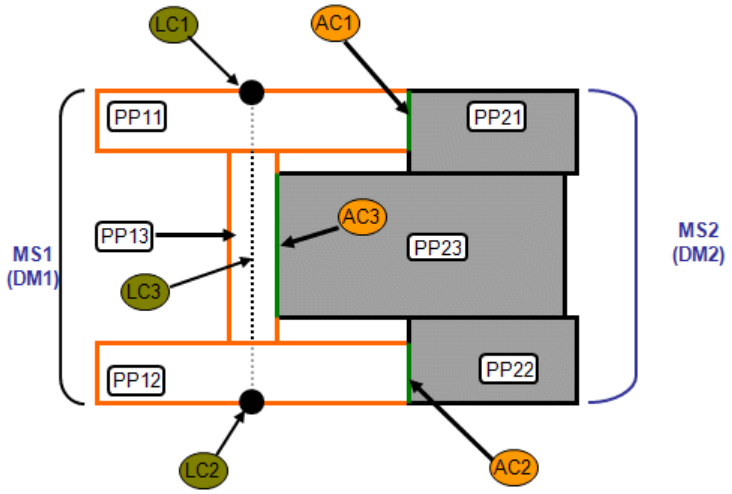
NOTE Object hierarchy is not shown.



Flange Thickness of Supported Member is Greater than Flange Thickness of Supporting Member

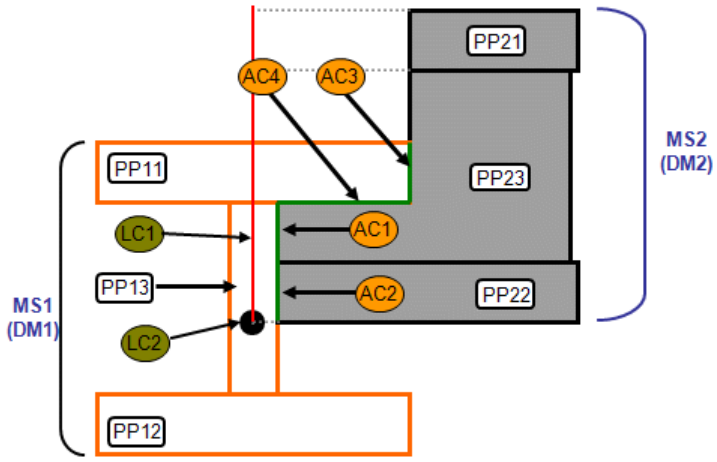
NOTES

- Object hierarchy is not shown.
- The width of web PP23 remains constant in this case.



Differing Control Points

NOTE Object hierarchy is not shown.

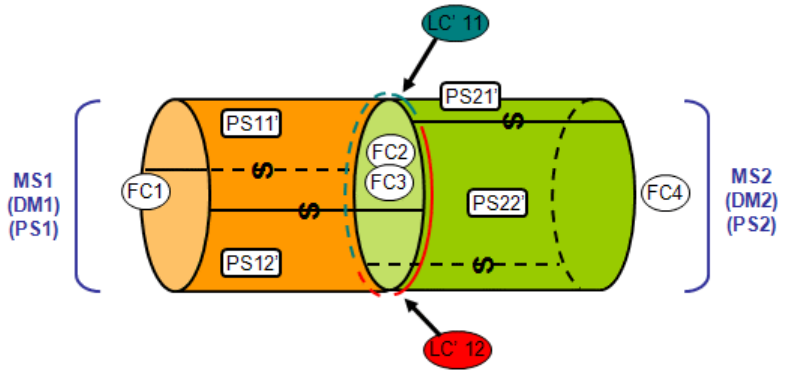


Tube End to Tube End - Co-Linear Axis

Member1(DM1)[Tube] then continue to Member2 (DM2) [Tube] OD Different, ID Same

Object Hierarchy Before Detailing

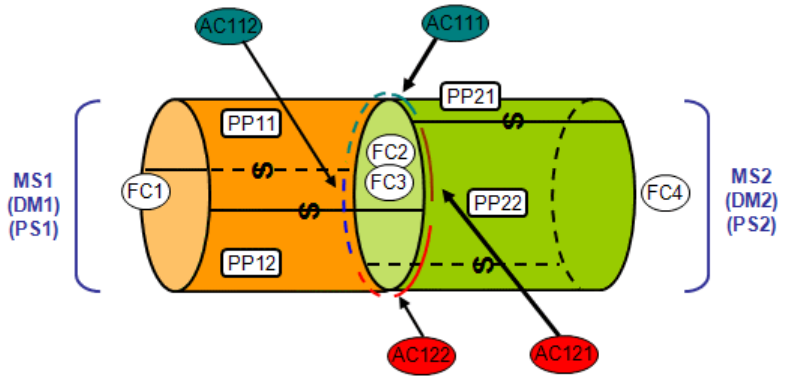
```
MS1
  - DM1
    - PS1
      - PS11'
        - PP11
      - PS12'
        - PP12
      - Seam1
      - Seam2
  -FC1
  -FC2
MS2
  - DM2
    - PS2
      - PS21'
        - PP21
      - PS22'
        - PP22
      - Seam1
      - Seam2
      - LC 1 (PS2 <-> MS1)
        - LC' 11 (PS21' <-> MS1)
        - LC' 12 (PS22' <-> MS1)
  - FC3
  - FC4
```



Object Hierarchy After Detailing

```
MS1
  - DM1
    - PS1
      - PS11'
        - PP11
      - PS12'
        - PP12
      - Seam1
      - Seam2
  -FC1
  -FC2
MS2
  - DM2
    - PS2
      - PS21'
        - PP21
      - PS22'
        - PP22
      - Seam1
      - Seam2
      - LC 1 (PS2 <-> MS1)
        - LC' 11 (PS21' <-> MS1)
          - AC111 (PP21 <-> PP11)
            - PC111
          - AC112 (PP21 <-> PP12)
            - PC112
        - LC' 12 (PS22' <-> MS1)
          - AC121 (PP22 <-> PP11)
            - PC121
          - AC122 (PP22 <-> PP12)
            - PC122
  - FC3
```

```
  - FC4
```



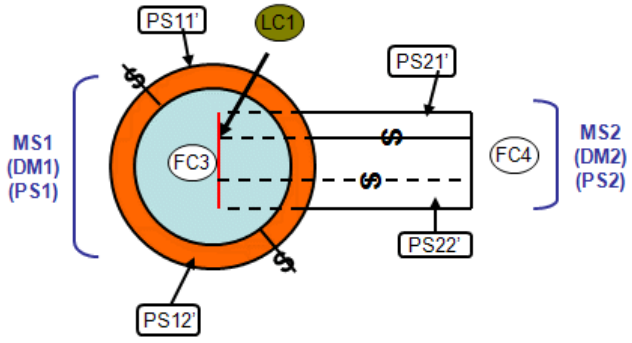
Tube to Tube - Axis Along

Member1(DM1)[Tube] connected to Member2 (DM2) [Tube/Cone] Axis Along Member2 into the side of Member1

NOTE Closed surfaces, such as cones and tubes, must have at least two longitudinal design splits for the software to detail them.

Object Hierarchy Before Detailing

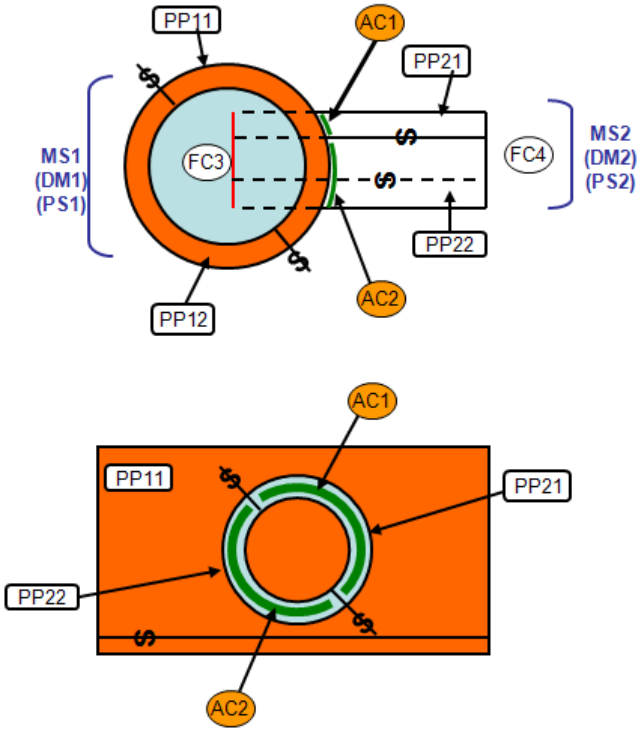
```
MS1
  - DM1
    - PS1
      - PS11'
        - PP11
      - PS12'
        - PP12
      - Seam1
      - Seam2
  -FC1
  -FC2
MS2
  - DM2
    - PS2
      - PS21'
        - PP21
      - PS22'
        - PP22
      - LC 1 (PS2 <-> MS1)
        - LC' 11 (PS21'<-> MS1)
        - LC' 12 (PS22'<-> MS1)
  - FC3
  - FC4
```



Object Hierarchy After Detailing

```
MS1
  - DM1
    - PS1
      - PS11'
        - PP11
      - PS12'
        - PP12
      - Seam1
      - Seam2
  -FC1
  -FC2
MS2
  - DM2
    - PS2
      - PS21'
        - PP21
      - PS22'
        - PP22
      - LC 1 (PS2 <-> MS1)
        - LC' 11 (PS21'<-> MS1)
          - AC1
            - PC1
        - LC' 12 (PS22'<-> MS1)
          - AC2
            - PC2
  - FC3
```

- FC4



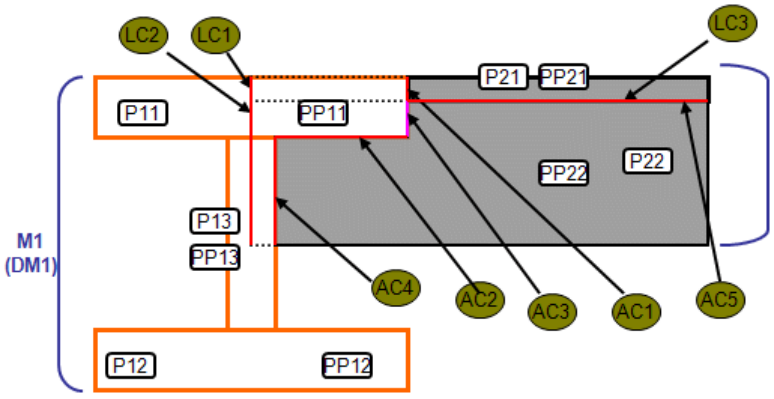
See Also

Detailing Built Up Connections (on page 993)

Designed Member to Designed Member

Member2(DM2) to Member1 (DM1) Axis Along (Flush on top flange, DM2 smaller depth and smaller flange thickness)

NOTE The frame connection is the same as in .*Designed Member to Designed Member - Axis Along* (on page 994)



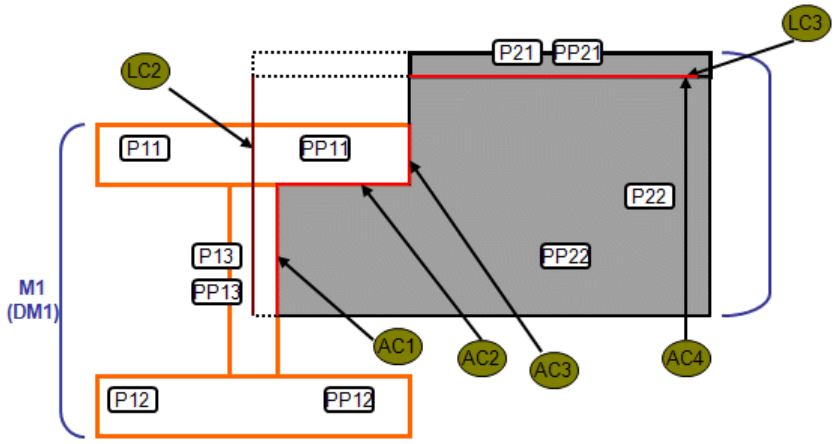
See Also

Detailing Built Up Connections (on page 993)

Designed Member to Designed Member - Axis Along

Member (DM2) to Member (DM1) Axis Along (M2 offset above M1)

NOTE The frame connection is the same as *Designed Member to Designed Member - Axis Along* (on page 994).



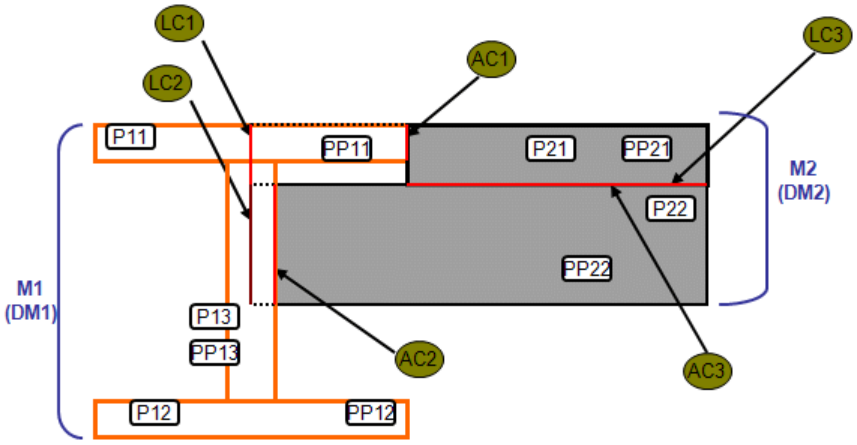
See Also

Detailing Built Up Connections (on page 993)

Designed Member to Designed Member - Axis Along

Member(DM2) to Member (DM1) Axis Along (Flush on top flange, M2 flange thickness > M1 flange thickness)

NOTE The frame connection is the same as *Designed Member to Designed Member - Axis Along* (on page 994).

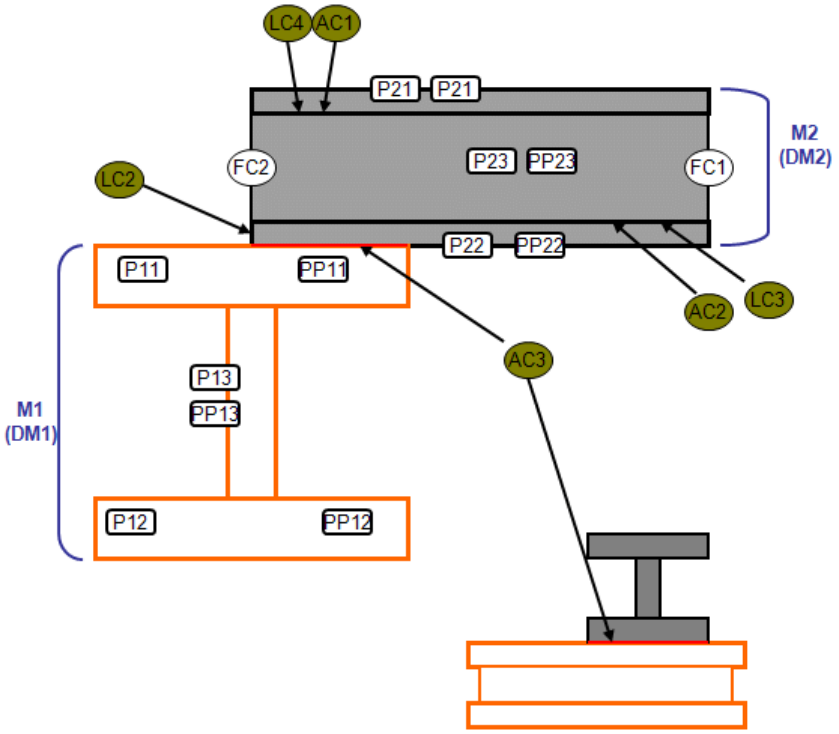


See Also

Detailing Built Up Connections (on page 993)

Designed Member to Designed Member - I Seated

Member2 (DM2) to Member1 (DM1) Axis Along (“I-Seated”)



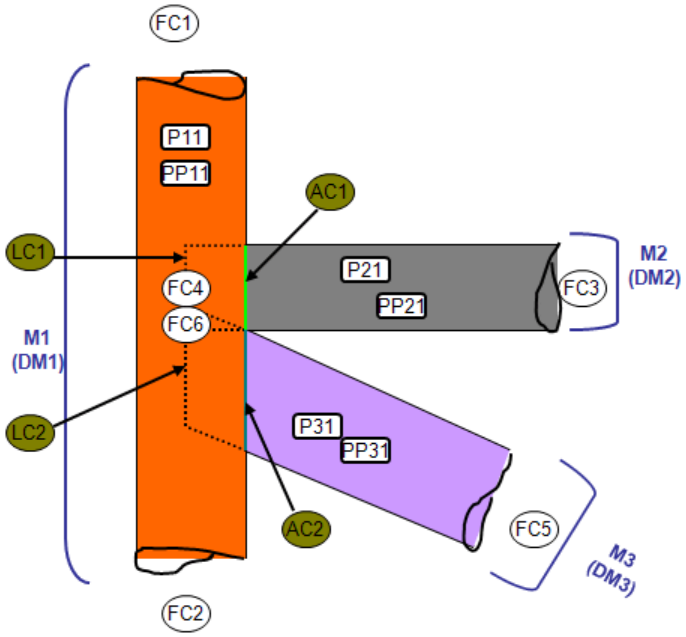
See Also

Detailing Built Up Connections (on page 993)

Vertical Brace or Gap

Vertical Brace or Gap 1 or Gap2

NOTE For Gap2, FC4 and FC6 are Gap2.

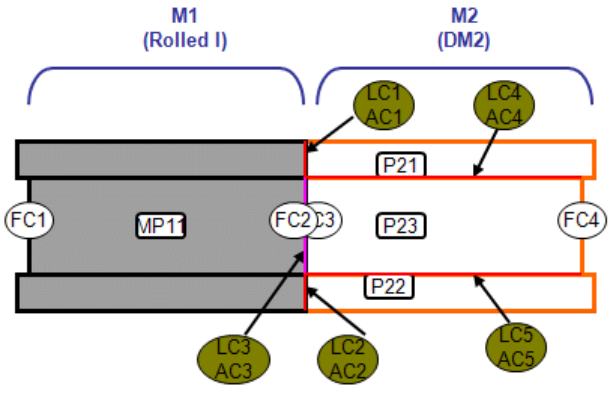


See Also

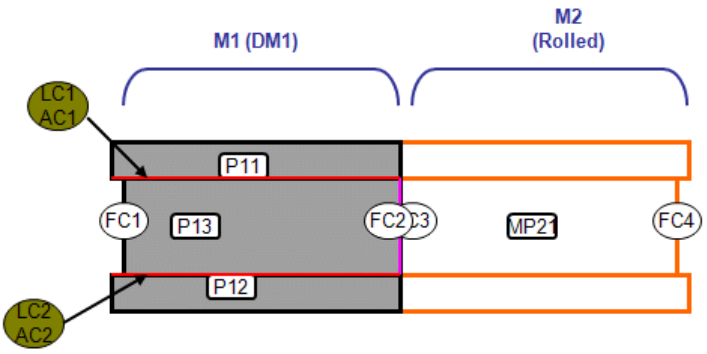
Detailing Built Up Connections (on page 993)

Rolled I to Built Up I

Member 1(Rolled-I) then continue to Member 2 (DM2) [BuiltUp-I]



Member 1(DM2) [BuiltUp-I] then continue to Member 2 (Rolled-I)

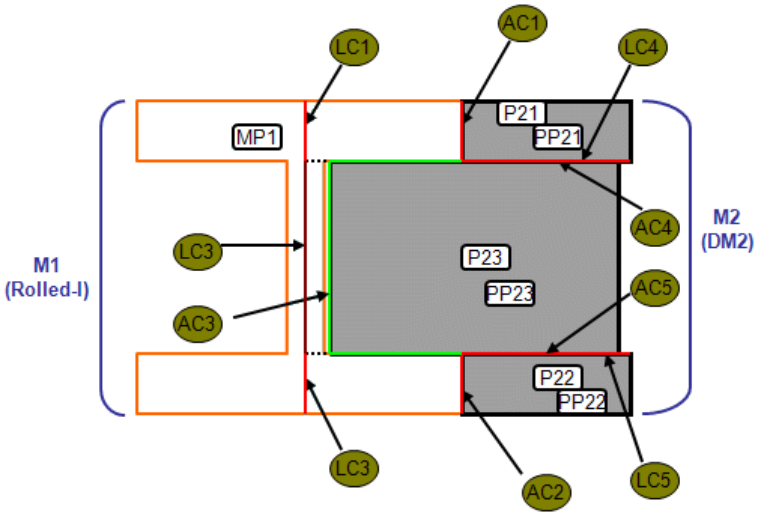


See Also

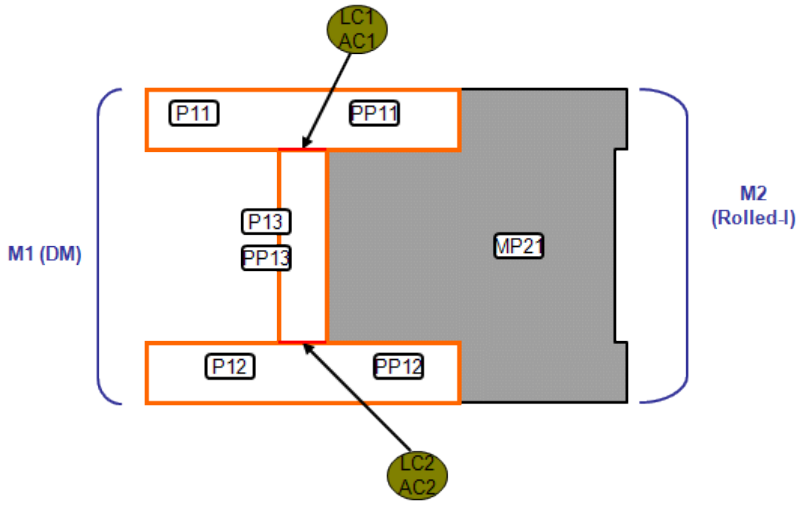
Detailing Built Up Connections (on page 993)

Designed Member to Rolled I

Member2 (DM2) to Member1 (Rolled - I) Axis Along

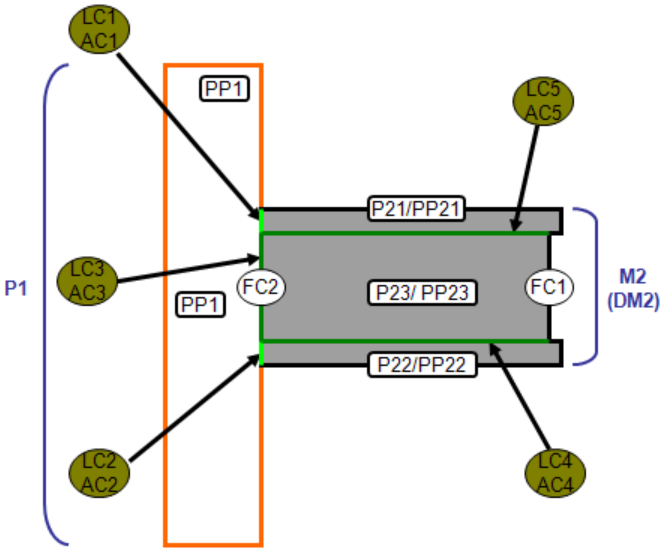


Member2(Rolled-I) to Member1 (DM1) Axis Along



Designed Member to Plate

Member2 (DM2) to Plate (P1) {Surface Connection}

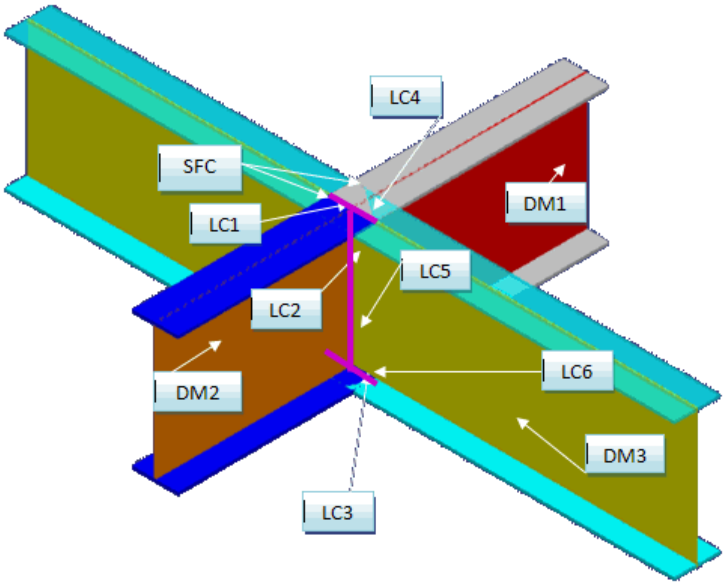


See Also

Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - 90° Center

DM1 and DM2 were created by splitting the original design member by DM3. Both members were defined using cardinal point 5.

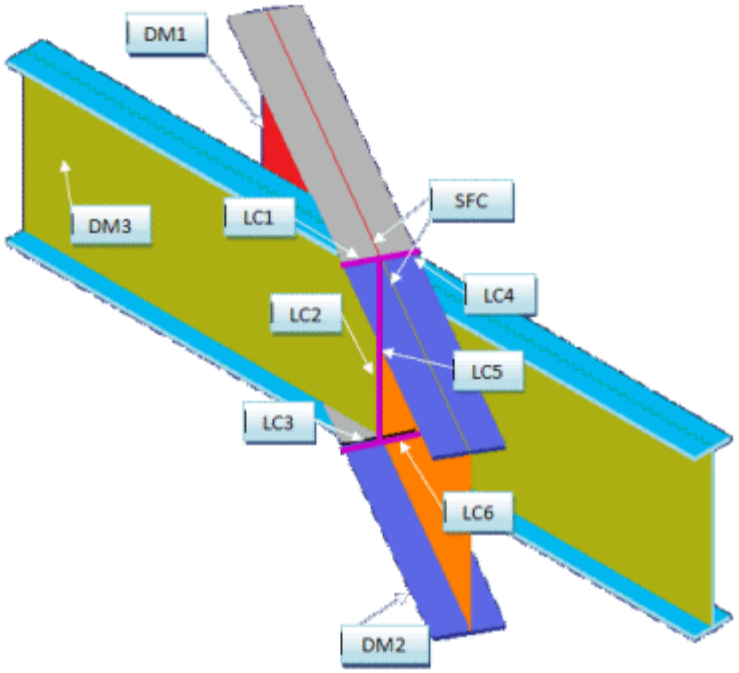


See Also

Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - Non Perpendicular Center

DM1 and DM2 were created by splitting the original design member by DM 3. Both members were defined using cardinal point 5.

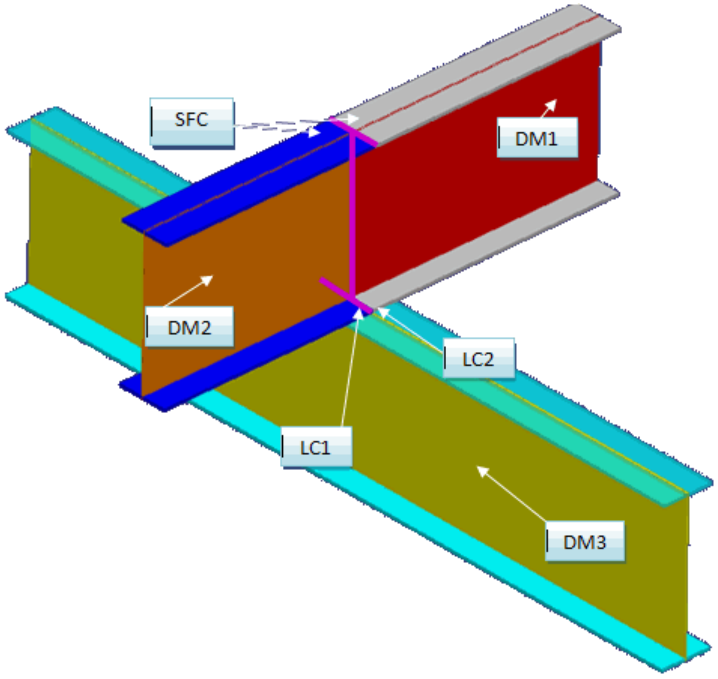


See Also

Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - 90º Opposite Connect Points

DM1 and DM2 were created by splitting the original design member by DM3. DM1 and DM2 use cardinal point 2. DM3 uses cardinal point 8.



NOTES

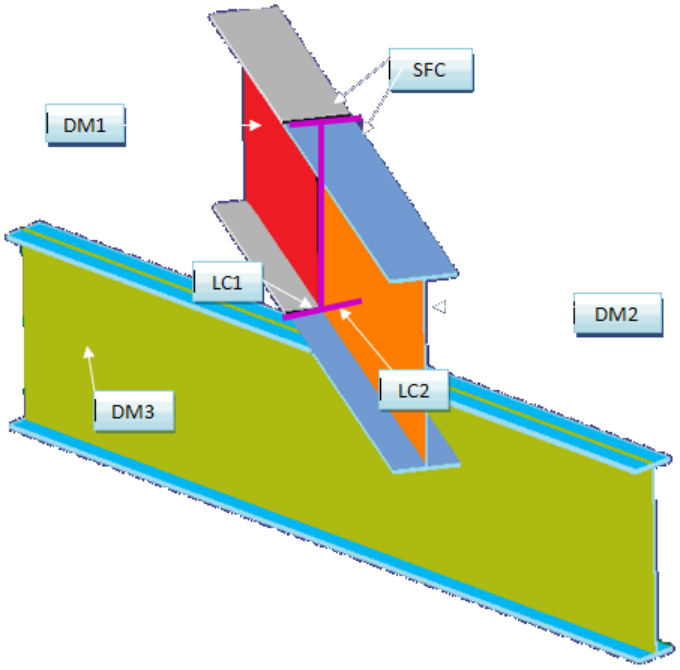
- This configuration is modeled with DM1 and DM2 using cardinal point 2 and DM3 using cardinal point 8. There is not a frame connection between the crossing members before the split. The member axes must intersect for the software to create the split. If there is a gap between the two crossing systems, the software cannot create the logical connections. Structural Detailing requires a connection to detail this case. The lapped property on the structural framing connection is sufficient for this.
- The orientation of the split is determined by the plane through the intersection point of the two member axes. This plane is normal to the split member axis.
- Structural Detailing does not create a connection between DM1 and DM2.
- If you place manual logical connections between the plate systems for DM1 and DM2, then assembly connections and physical connections are created as children of the manual logical connections.

See Also

Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - Non Perpendicular Opposite Connect Points

DM1 and DM2 were created by splitting the original design member by DM3. DM1 and DM2 use cardinal point 2. DM3 uses cardinal point 8.



NOTES

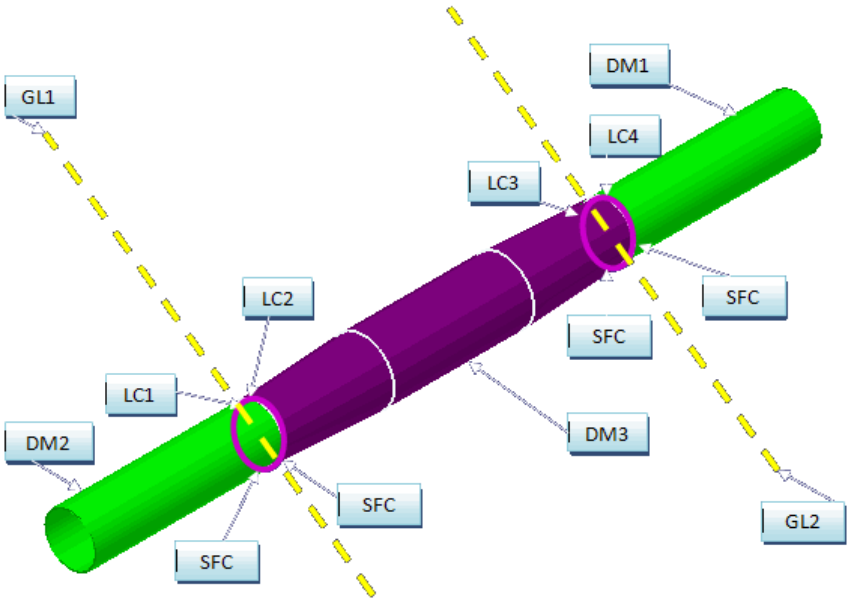
- The orientation of the split is not modified by detailing. The trim of the lower flanges for DM1 and DM2 are normal to the direction of the member.
- If manual logical connections are placed between plate systems for DM1 and DM2, then assembly connections and physical connections are created as children of the manual logical connections.

See Also

Detailing Built Up Connections (on page 993)

Split Built Up by Grid - Perpendicular

DM2, DM2, and DM3 were created by splitting the original design member by GL1 and GL2.



NOTES

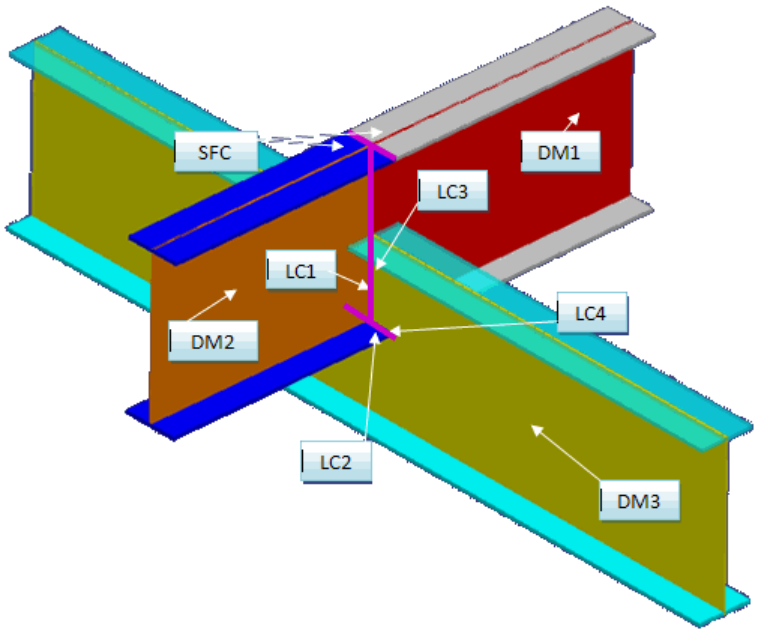
- Assembly connections and physical connections are created as children of the logical connections.
- The software checks to ensure than duplicate assembly connections and physical connections are not created. If duplicate logical connections exist, the assembly connections, physical connections, and chamfers are created as children of the first logical connection processed.

See Also

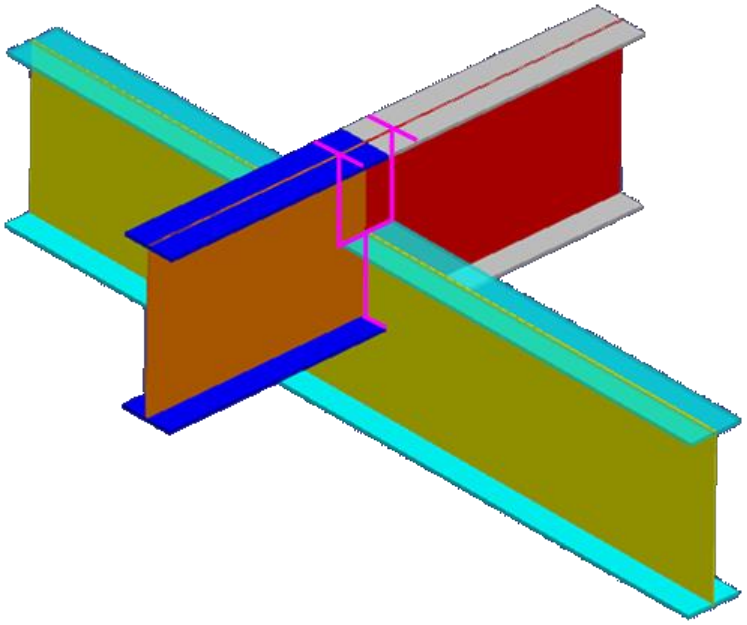
Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - 90º Overlapping Connect Points

DM1 and DM2 were created by splitting the original design member by DM3.



The following figure indicates the trim lines after detailing:

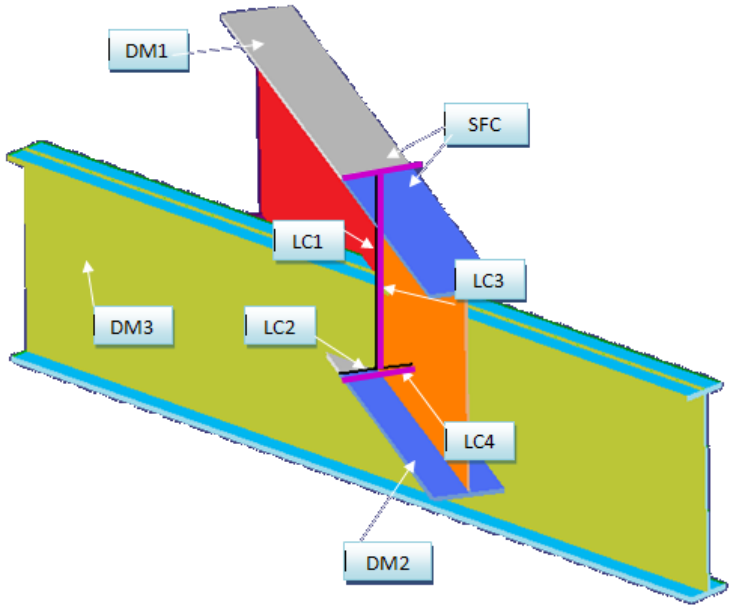


See Also

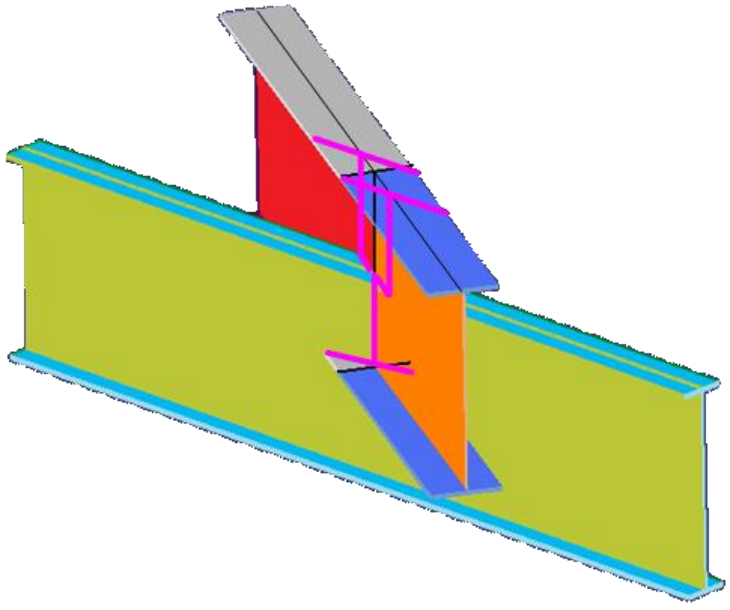
Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - Non Perpendicular Partial Penetration

DM1 and DM2 were created by splitting the original design member by DM3.



The following figure indicates the trim lines after detailing:

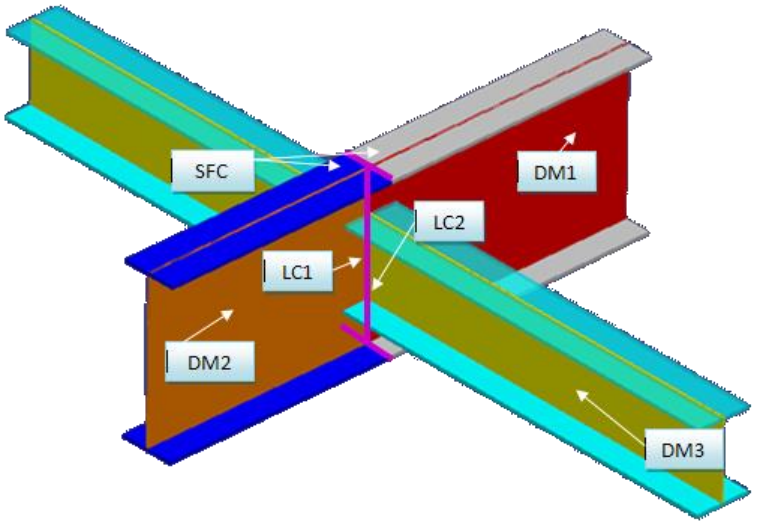


See Also

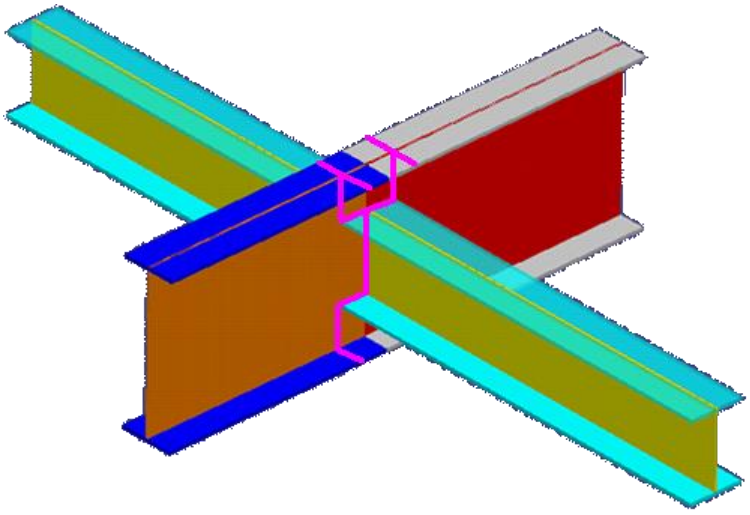
Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - 90º Penetration

DM1 and DM2 were created by splitting the original design member by DM3.



The following figure indicates the trim lines after detailing:

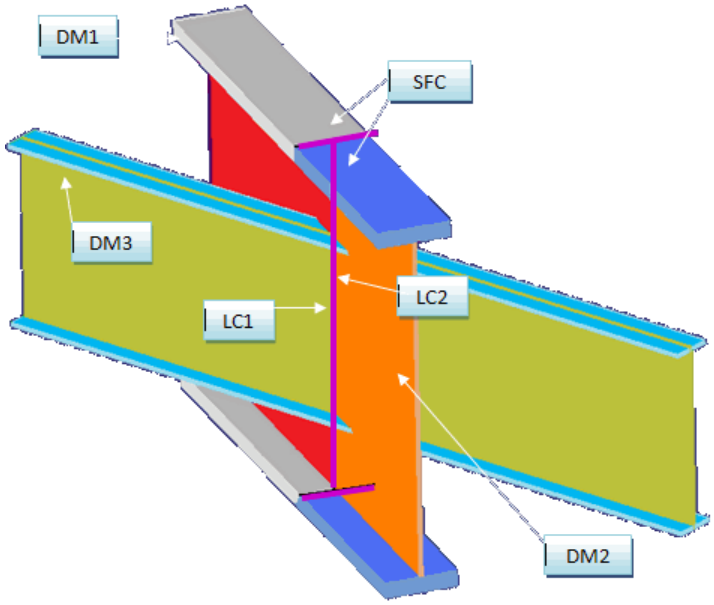


See Also

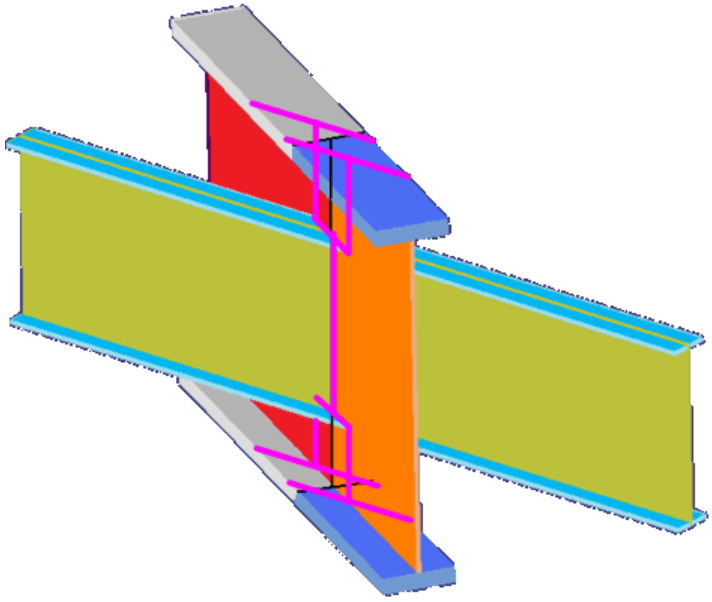
Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - Non Perpendicular Penetration

DM1 and DM2 were created by splitting the original design member by DM3. Both members are defined using cardinal point 5.



The following figure indicates the trim lines after detailing:

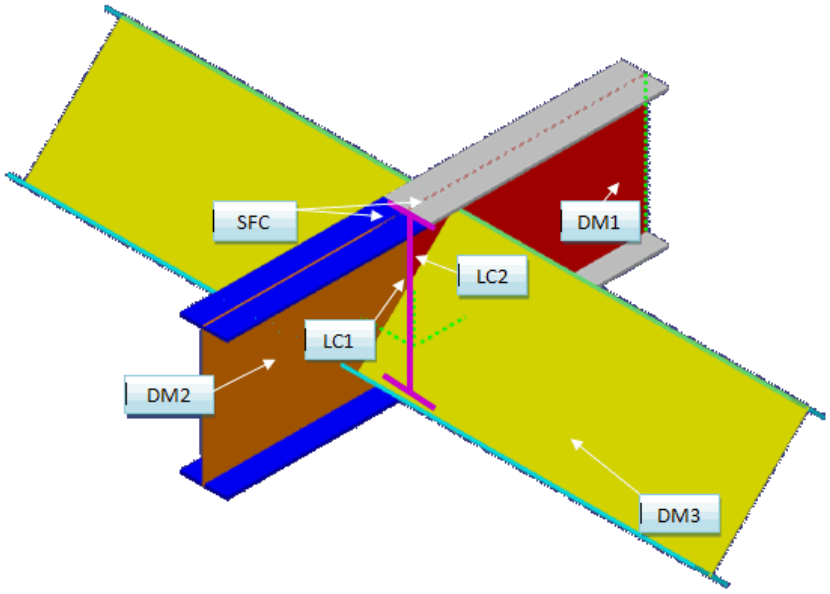


See Also

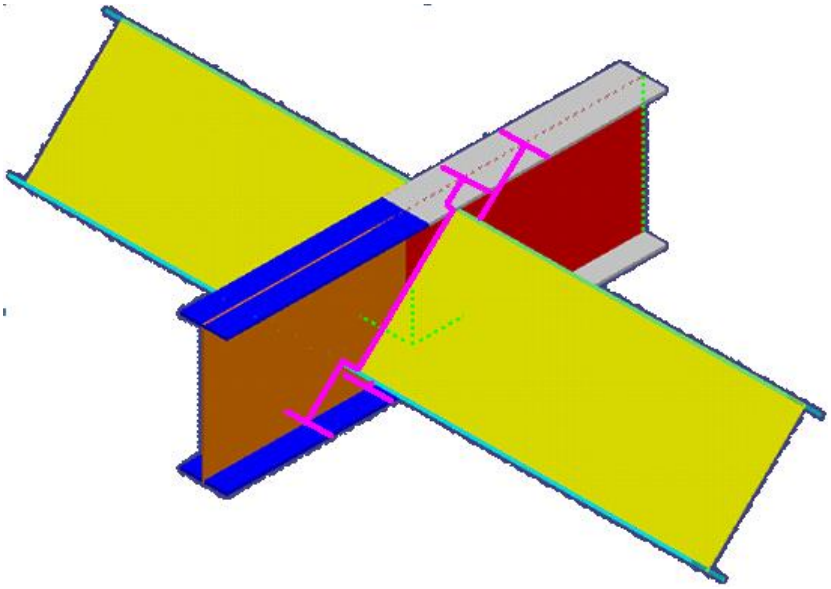
Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - 90º Rotation of Member

DM1 and DM2 were created by splitting the original design member by DM3. Both members are defined using cardinal point 5.



The following figure indicates the trim lines after detailing:

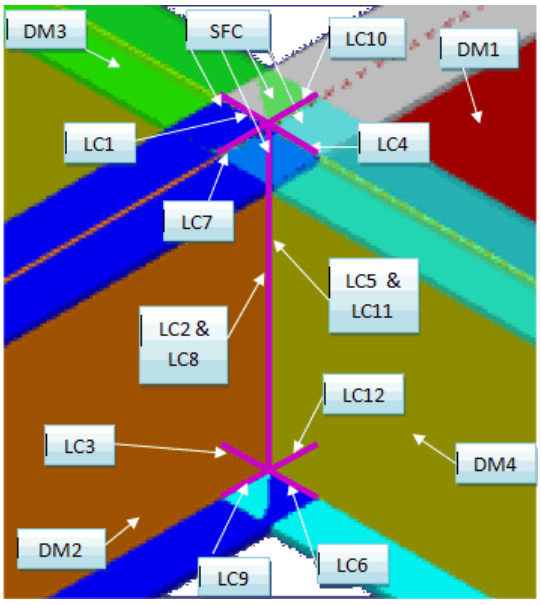


See Also

Detailing Built Up Connections (on page 993)

Split Built Up by Built Up - 90º Split Both

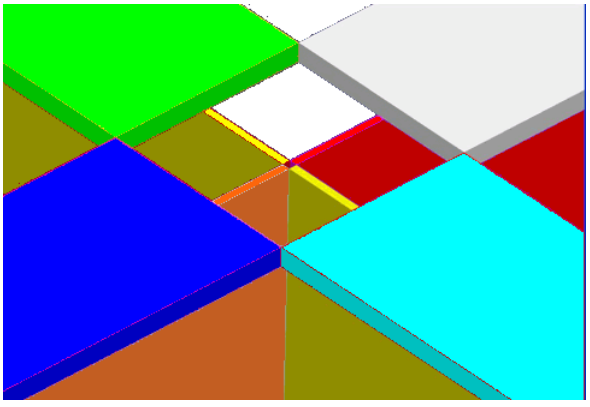
DM1 and DM2 were created by splitting the original design member by DM3 and DM4. DM3 and DM4 were created by splitting the original design member by DM1 and DM2.



NOTES

- Structural Detailing creates only the split frame connections. This is an ambiguous case that you need to modify later in the design cycle with additional objects such as column or side plates, so logical connections are not created.
- All assembly and physical connections are placed on the **To Do List** because there is no physical contact after trimming.

The following figure indicates the trim lines after detailing:



See Also

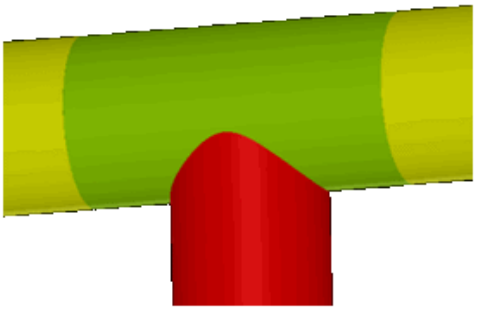
Detailing Built Up Connections (on page 993)

Can - Horizontal Beam

The beam is horizontal and continuous before the can is placed.

NOTES

- The beam can be a built-up member.
- The beam can be a standard member.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.



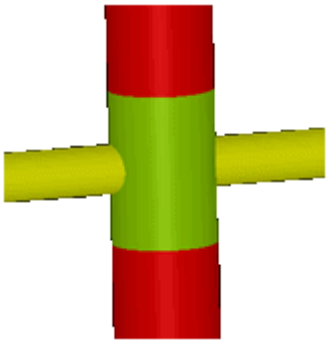
See Also

Detailing Built Up Connections (on page 993)

Can - Vertical Column

NOTES

- The column is continuous before the can is placed.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.



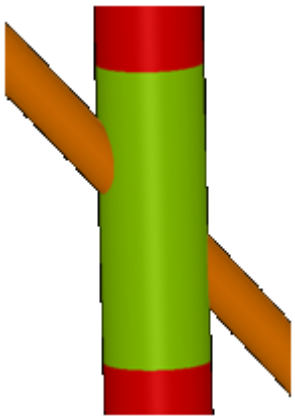
See Also

Detailing Built Up Connections (on page 993)

Can - Continuous Leg

NOTES

- The leg is continuous.
- The column is doubly sloping (sloping X-Z and Y-Z plane).
- The column is continuous before the can is placed.
- The software does not support an incoming continuous leg. The leg must be bound to the column that has the can inserted.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.



See Also

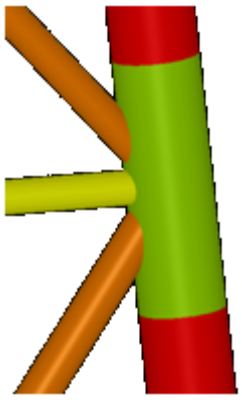
Detailing Built Up Connections (on page 993)

Can - Multiple Braces

NOTES

- The column is doubly sloping (sloping X-Z and Y-Z plane).
- The column is continuous before the can is placed.
- The beam is a built-up member.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.



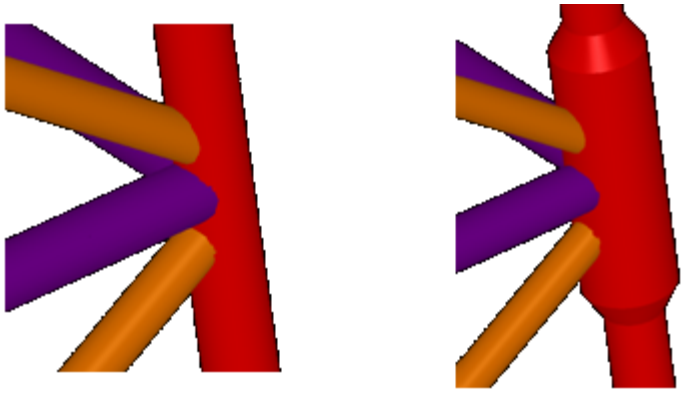
See Also

Detailing Built Up Connections (on page 993)

Can - Diameter Change

NOTES

- The column is doubly sloping (sloping X-Z and Y-Z plane).
- The column is continuous before the can is placed.
- The can is placed with an outer diameter that is larger than either leg.
- The beam is a built-up member.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.



See Also

Detailing Built Up Connections (on page 993)

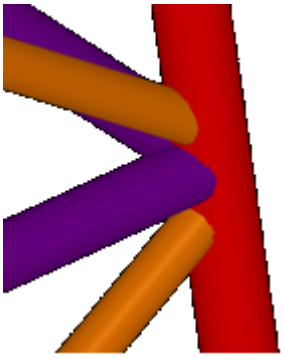
Can - Single Axis Gap

NOTES

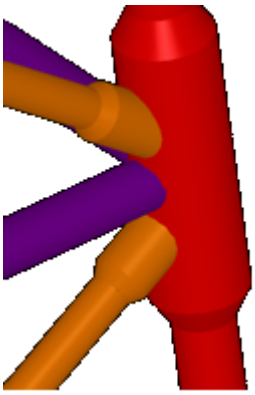
- The column is doubly sloping (sloping X-Z and Y-Z plane).
- The column is continuous before the can is placed.
- Rule-based gaps are applied (AxisGap - Single) to the vertical braces shown in orange before the can is placed.
- The vertical braces and beams are all coplanar.
- The beam is a built-up member.
- The can is placed with an outer diameter larger than either leg.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.

Two Vertical Braces with Gap Applied

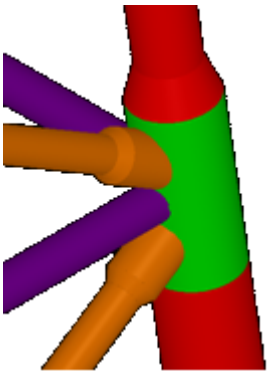
The vertical braces are coplanar. They are smaller than the can and are built-up tubes.



Stub End Cans Added to Vertical Braces



Bottom Leg Matches the Outer Diameter of the Can



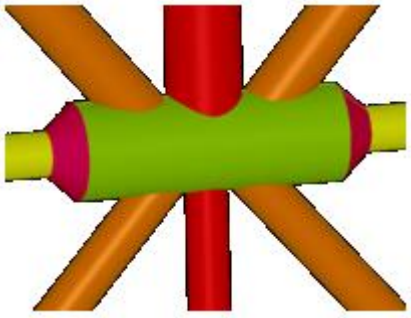
See Also

Detailing Built Up Connections (on page 993)

Can - Multiple Braces

NOTES

- The beam is continuous before the can is placed. Only the columns (shown in red) exist on either side.
- An oversized can is applied to the beam (shown in yellow).
- Braces are added to the can with rule based gaps (AxisGap - Single).
- The vertical braces, columns, and beams are all coplanar.
- The cone is set to have a matching ID with the can.
- Members are built-up members.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.



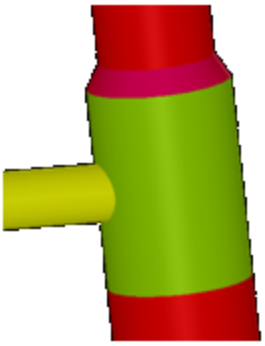
See Also

Detailing Built Up Connections (on page 993)

Can - Cone Transition

NOTES

- The bottom leg and top leg are of different sizes.
- The can matches the outer diameter of the bottom leg.
- The cone matches the outer diameter of the top leg.
- The column is continuous before the can is placed.
- The beam is a built-up member
- The can is inserted first, then the leg sizes are changed.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.



See Also

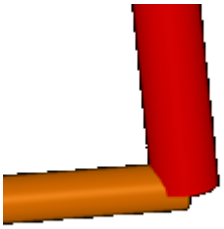
Detailing Built Up Connections (on page 993)

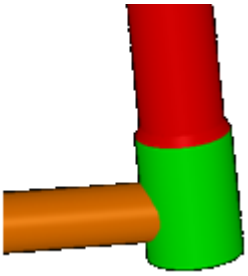
Can - Beam and Column End Point

NOTES

- The beam and the column share the same end point.
- The can has a larger outer diameter than the outer diameter of the leg.
- The beam is a built-up member.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.





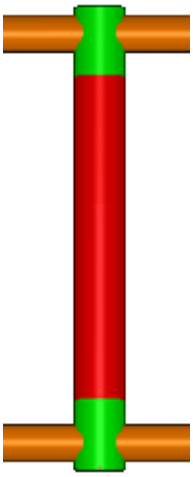
See Also

Detailing Built Up Connections (on page 993)

Can - Beam and Column Intersection

NOTES

- The can matches the outer diameter and ID of the column.
- The beam is horizontal and split before the can is placed.
- For more information on the chamfer properties, see Chamfer Properties Dialog Box.
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.



See Also

Detailing Built Up Connections (on page 993)

SECTION 17

Geometric Constructions

Geometric constructions are geometry, such as points, curves, and surfaces, created directly in the 3D graphic view. The software used the geometry to create objects in the model, such as plates, seams, knuckles, and physical connections. The geometry is typically used in a progressive manner, such as:

- Create points
- Create curves using the points
- Create a surface using the curves
- Create a plate system using the surface

The **Geometric Construction Palette** displays when you click the option on the ribbon of a supported command. Each tab of the dialog box graphically displays macros in the catalog database.

Geometric Construction Palette

IMPORTANT Each command using the option displays the palette with dialog box tabs and geometric construction icons appropriate to the command.

Filters

Limits the available macros on a tab to the selected type.

Clear Recent

Removes recently selected macros from the **Recent** tab.

Help

Displays help for the selected macro in a preview window. If no help is available, a preview of the selected macro displays.

NOTE You can also select **Tools** > **Automatic Preview** to display the preview window.

The **Geometric Construction Palette** typically contains the following tabs:

- **Points** - Displays interfaces for creating points.
- **Curves** - Displays interfaces for creating lines and curves. Some related point interfaces also appear.
- **Surfaces** - Displays interfaces for creating planes and surfaces. Some related curve and point interfaces also appear.
- **Coordinate Systems** - Displays interfaces for creating coordinate systems.
- **Recent** - Displays geometric construction interfaces recently selected on the other tabs. Click **Clear Recent** to remove interfaces from this tab.

The following geometric construction types are available. These interfaces appear in different combinations on the **Geometric Construction Palette** tabs.

Points (Geometric Construction Palette Dialog Box) (on page 1028)

Curves (Geometric Construction Palette Dialog Box) (on page 1061)

Surfaces (Geometric Construction Palette Dialog Box) (on page 1115)

Coordinate Systems (Geometric Construction Palette Dialog Box) (on page 1168)

Solids (Geometric Construction Palette Dialog Box) (on page 1176)

After you select an interface, the ribbon for that interface displays.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

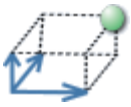
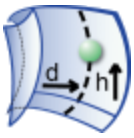
The **Geometric Construction Explorer** dialog box also displays when is clicked. As geometry is created, a hierarchical list of applicable surfaces, curves, and points display.

Geometric Construction Explorer **(on page 1214)**

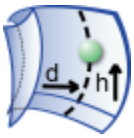
Points (Geometric Construction Palette Dialog Box)

The following geometric construction interfaces create points.

NOTE Each command supporting the option displays tabs and interfaces appropriate to the command.

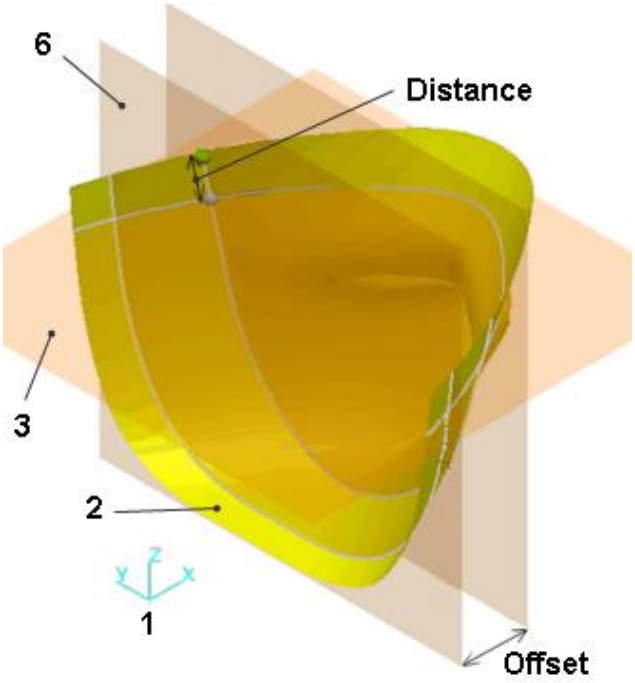
	**Points Along Curve** - Creates evenly-spaced points along a curve. For more information, see *Points Along Curve* (on page 1047).
	**Point from Coordinate System** - Creates a point at the X, Y, and Z coordinates of the selected coordinate system. For more information, see *Point from Coordinate System* (on page 1039).
	**Point Along Curve on Surface** - Creates a point along a curve on a surface relative to the selected coordinate system. For more information, see *Point Along Curve on Surface* (on page 1029).
	**Vertices from Curve or Surface** - Creates points at the vertices of a curve or surface. For more information, see *Vertices From Curve or Surface* (on page 1057).

Point Along Curve on Surface



Tooltip: PointAlongCurveOnSurface

Creates a point along a curve on a surface relative to the selected coordinate system.



Required Input	**Output**
1 - Coordinate system	Point along the curve
2 -Surface	
3 - Reference plane intersecting the surface	
6 - Supporting plane intersecting the surface	
Optional Input	
Points or curves to define reference plane **3**	
Parameters	
Offset method and value	
Distance method and value	

Point Along Curve on Surface Ribbon (for geometric constructions)

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the coordinate system. The coordinate system is used when there is an ambiguous solution.

2

Specifies the surface. By default, this is the surface of the plate selected for the main command.

3

Specifies a reference plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the reference curve from which the new point is offset as defined by **Offset** and **Offset Direction**.

(4)

Specifies a point used to define the plane intersecting the surface. This option is only available if a point or curve is selected for **3**.

5

Specifies a third point used to define the plane intersecting the surface. This option is only available if points are selected for **3** and 4.

6

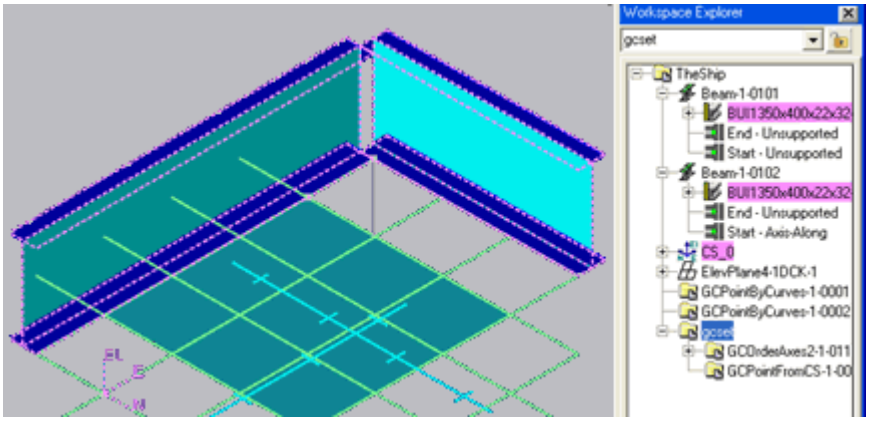
Specifies a supporting plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the supporting curve along which the new point is located as defined by **Distance** and **Distance Method**.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

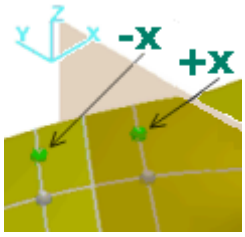
★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Offset

Defines the offset distance of the point along the reference curve.

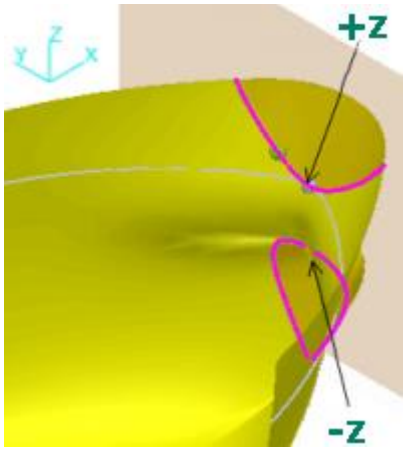
Offset Direction

Defines the offset direction (with respect to the selected coordinate system **1**) along the reference curve. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available when **Offset** is set to a value other than **0**.



Inter Ambiguity

Specifies the solution when the supporting plane intersects a complex surface in multiple locations. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. The following example shows the ambiguity options in the bow of a hull.



Distance

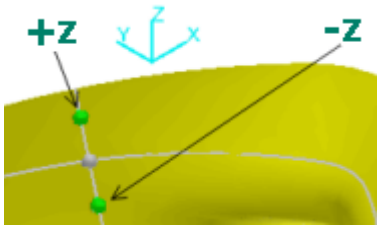
Defines the distance of the point from the reference curve along the supporting curve.

Distance Method

Defines how the point is measured along the supporting curve. Select **Girth** (on page 1457), **Along x**, **Along y**, or **Along z**.

Point Ambiguity

Specifies the solution along the supporting curve when **Distance** is not **0** and **Girth** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Girth**.

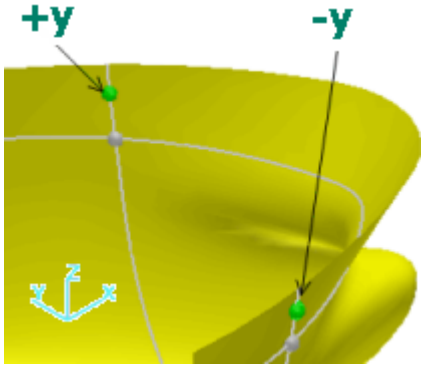


Distance Direction

Specifies the solution along the supporting curve when **Distance** is not **0** and **Along x**, **Along y**, or **Along z** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Along x**, **Along y**, or **Along z**.

Side Ambiguity

Specifies the solution when the reference plane intersects a complex surface in more than one location, such as on the port and starboard sides of a hull. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**.



WARNING For options that have coordinate axis values, select a value that is compatible with the orientation of the curve. For example, if a non-linear reference curve is in the x-y plane, valid options for **Offset Direction** are **+x**, **-x**, **+y**, and **-y**.

Point Along Curve on Surface Ribbon (for topological points in the Molded Forms task)

Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming

Resources is installed.

1

Specifies the coordinate system. The coordinate system is used when there is an ambiguous solution.

2

Specifies the surface. By default, this is the surface of the plate selected for the main command.

3

Specifies a reference plane intersecting the surface, or a curve or point used to define the plane. The curve created by this intersection is the reference curve from which the new point is offset as defined by **Offset** and **Offset Direction**.

(4)

Specifies a point used to define the plane intersecting the surface. This option is only available if a point or curve is selected for **3**.

(5)

Specifies a third point used to define the plane intersecting the surface. This option is only available if points are selected for **3** and (4).

6

Specifies a supporting plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the supporting curve along which the new point is located as defined by **Distance** and **Distance Method**.

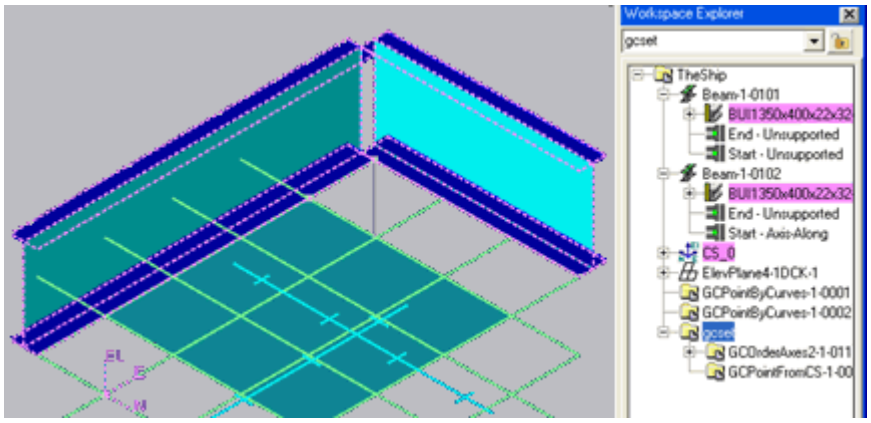
Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background when no input is selected.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- 1 and 0.00 m - No background when the value of the original geometric construction is retained.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

- 1 - Blue background when the input is the same for all selected geometric constructions.
- 1 - No background with pink underline when the input is different for *all* selected geometric constructions.
- 1 - Blue background with pink underline when the input is different for *some* selected geometric constructions.
- 1 - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.
- - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.



Reject

Clears the current selection.

Accept

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.
- **More** - Select a system in the model.

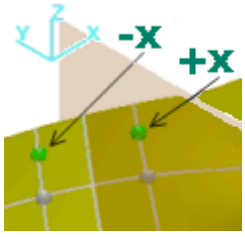
NOTE The **New GC Set** option is not used for topological points.

Offset

Defines the offset distance of the point along the reference curve.

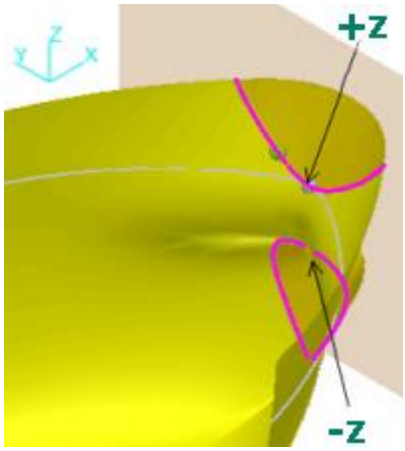
Offset Direction

Defines the offset direction (with respect to the selected coordinate system **1**) along the reference curve. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available when **Offset** is not set to **0**.



Inter Ambiguity

Specifies the solution when the supporting plane intersects a complex surface in multiple locations. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. The following example shows the ambiguity options in the bow of a hull.



Distance

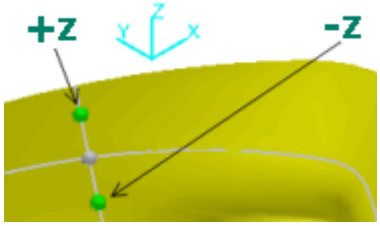
Defines the distance of the point from the reference curve along the supporting curve.

Distance Method

Defines how the point is measured along the supporting curve. Select **Girth** (on page 1457), **Along x**, **Along y**, or **Along z**.

Point Ambiguity

Specifies the solution along the supporting curve when **Distance** is not **0** and **Girth** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Girth**.

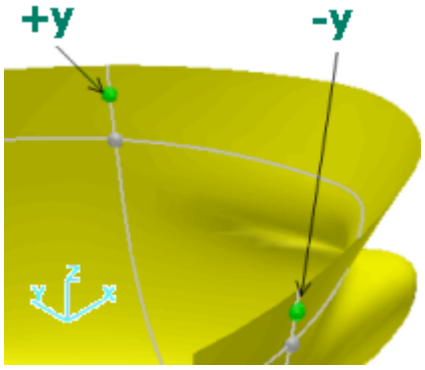


Distance Direction

Specifies the solution along the supporting curve when **Distance** is not **0** and **Along x**, **Along y**, or **Along z** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Along x**, **Along y**, or **Along z**.

Side Ambiguity

Specifies the solution when the reference plane intersects a complex surface in more than one location, such as on the port and starboard sides of a hull. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**.



IMPORTANT For options that have coordinate axis values, select a value that is compatible with the orientation of the curve. For example, if a non-linear reference curve is in the x-y plane, valid options for **Offset Direction** are **+x**, **-x**, **+y**, and **-y**.

What do you want to do?

- *Place a point on a surface and along a curve* (on page 1038)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)
- *Place a topological point on a surface and along a curve* (on page 1333) (in the Molded Forms task)
- *Modify a topological point* (on page 1334) (in the Molded Forms task)
- *Change a topological point to a different point definition* (on page 1335) (in the Molded Forms task)
- *Delete a topological point* (on page 1335) (in the Molded Forms task)

Place a point on a surface and along a curve

1. Click **PointAlongCurveOnSurface** on the **Geometric Construction Palette**.
2. Select a coordinate system.
3. Select a surface, such as a plate system or reference plane.

   NOTE If a surface was selected for the main command, then this is the default value and **2** has a yellow background. If you want to use this surface, skip this step and click **3**.
4. Select a plane intersecting the surface. The **Offset** direction is along the reference curve created by this intersection.
5. Select another plane intersecting the surface and the reference curve. The **Distance** direction is along the supporting curve created at the intersection with the surface.
6. Select a value for **Distance Method**.

7. Type the needed values for **Offset** and **Distance**.

   A preview point in green appears at the intersection of the surface, reference curve, and supporting curve. If more than one point appears, then more than one solution is possible.

8. If multiple points appear, perform one of the following steps to resolve the ambiguity:

   - If the surface is complex (a plane intersects it more than once), then select a coordinate axis direction for **Inter Ambiguity**.
   - If **Girth** is selected for **Distance Method**, then select a coordinate axis direction for **Point Ambiguity**.
   - If two solutions are possible on opposing sides of the reference curve, then select a coordinate axis direction for **Side Ambiguity**.

   NOTE For each ambiguity, select a value that is compatible with the orientation of the points.

   A single green preview point is now visible.

9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **[Not Named]** in the **Geometric Construction Explorer**.*



10. If needed, create additional geometric constructions.
11. Click **Close** to return to the main command ribbon.

IMPORTANT

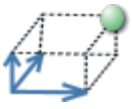
- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point from Coordinate System



Tooltip: PointFromCS

Creates a point at the defined X, Y, and Z distances from the origin of the selected coordinate system or from a selected point.

Required Input	**Output**
Coordinate system	Point
Optional Input	
Reference point	
Parameters	
X, Y, and Z distances	

Point from Coordinate System Ribbon (for geometric constructions)

 **Select**

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

 **Add**

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

 **Delete**

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click  again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the coordinate system.

(2)

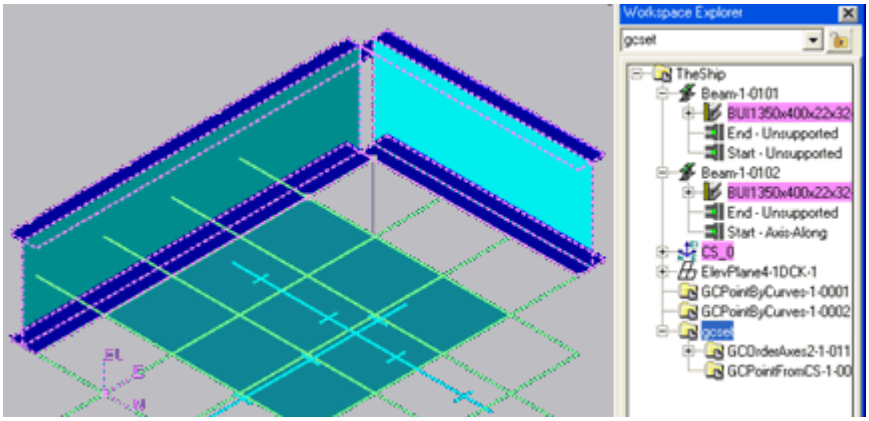
Specifies a reference point from which X, Y, and Z distances are measured. If no reference point is selected, the X, Y, and Z distances are measured from the origin of the coordinate system. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

X

Specifies the distance along the X-axis from the origin or reference point.

Y

Specifies the distance along the Y-axis from the origin or reference point.

Z

Specifies the distance along the Z-axis from the origin or reference point.

Point from Coordinate System Ribbon (for topological points in the Molded Forms task)

 **Properties**

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select a different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

Specifies the coordinate system.



Specifies a reference point from which X, Y, and Z distances are measured. If no reference point is selected, the X, Y, and Z distances are measured from the origin of the coordinate system. This value is optional.

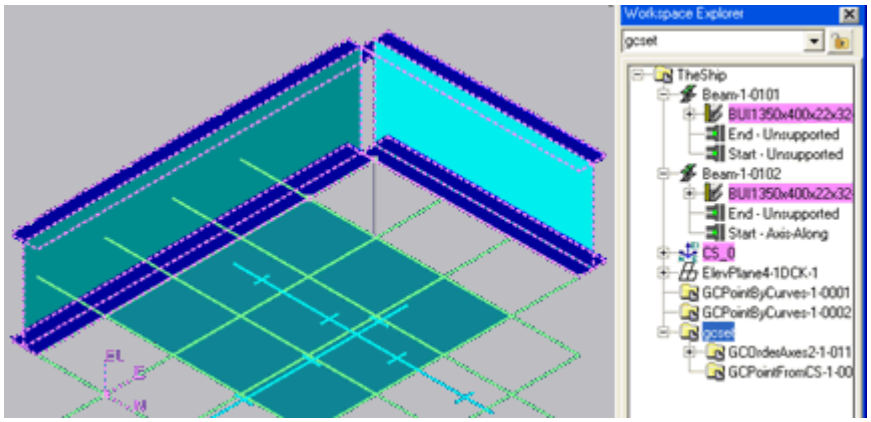
Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background when no input is selected.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- 1 and 0.00 m - No background when the value of the original geometric construction is retained.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

- 1 - Blue background when the input is the same for all selected geometric constructions.
- 1 - No background with pink underline when the input is different for *all* selected geometric constructions.
- 1 - Blue background with pink underline when the input is different for *some* selected geometric constructions.
- 1 - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.
- - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.



Reject

Clears the current selections.

Accept

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.
- **More** - Select a system in the model.

NOTE The **New GC Set** option is not used for topological points.

X

Specifies the distance along the X-axis from the origin or reference point.

Y

Specifies the distance along the Y-axis from the origin or reference point.

Z

Specifies the distance along the Z-axis from the origin or reference point.

What do you want to do?

- *Place a point by distances from a coordinate system origin* (on page 1045)
- *Place a point by distances from a reference point* (on page 1045)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)
- *Place a topological point by distances from a coordinate system origin* (on page 1364) (in the Molded Forms task)
- *Place a topological point by distances from a reference point* (on page 1364) (in the Molded Forms task)
- *Modify a topological point* (on page 1334) (in the Molded Forms task)
- *Change a topological point to a different point definition* (on page 1335) (in the Molded Forms task)
- *Delete a topological point* (on page 1335) (in the Molded Forms task)

Place a point by distances from a coordinate system origin

1. Click **PointFromCS** on the **Geometric Construction Palette**.
2. Select a coordinate system.
3. Type values for **X**, **Y**. and **Z**.

   *A green preview point appears. It is offset from the coordinate system origin by the **X**, **Y**. and **Z** values.*
4. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and in the **Geometric Construction Explorer**.*



5. If needed, create additional geometric constructions.
6. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place a point by distances from a reference point

1. Create the point to use as a reference point. The point can be created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **PointFromCS** on the **Geometric Construction Palette**.
3. Select a coordinate system.
4. Click (2), and select the reference point.
5. Type values for **X**, **Y**, and **Z**.

   *A green preview point appears. It is offset from the reference point by the **X**, **Y**, and **Z** values*

*and along the **X**, **Y**, and **Z** axes of the coordinate system.*

6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and in the **Geometric Construction Explorer**.*



7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

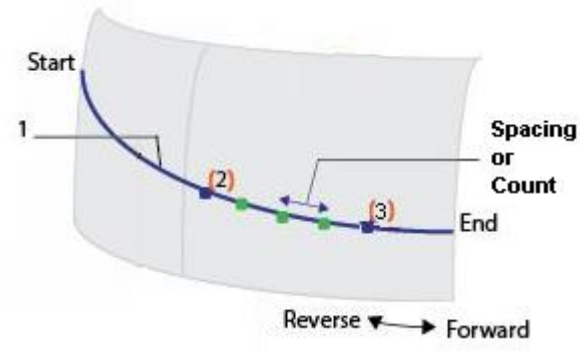
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Points Along Curve



Tooltip: PointsAlongCurve

Creates an evenly-spaced pattern of points along a curve. Points are placed by defining the spacing between points or the total number of points.



Required Input	Output
1 - Curve, line, or edge	Points
Optional Input	
(2) - Start point	
(3) - End point	
Parameters	
Point spacing or point count	

Points Along Curve Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the curve on which the points are placed.

(2)

Specifies the starting point for the point pattern. This value is optional.

(3)

Specifies the ending point for the point pattern. This value is optional.

(4)

Specifies a tracking point used to control the direction of point placement an ambiguity exists. A tracking point is only needed with some curves, such as:

- A circle defined by two points and a mid-point in which there are two arcs between the start and end points.
- A curve on multiple sides of a non-planar surface, such as a seam or reference curve wrapping around a hull.

This value is optional.

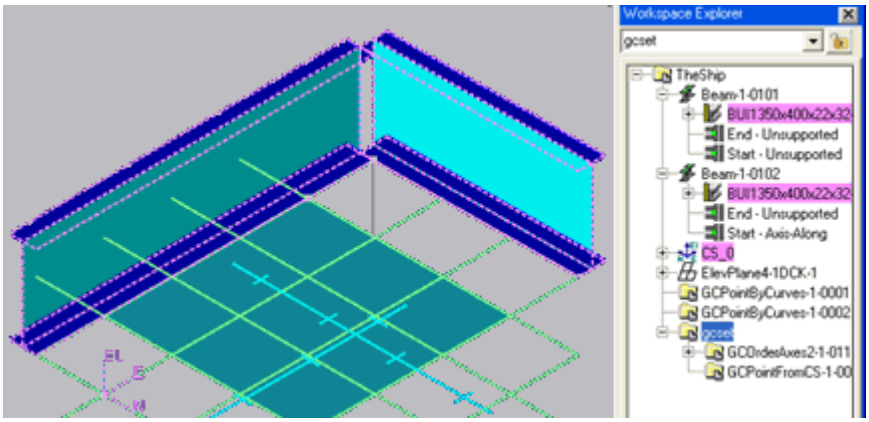
NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Direction

Specifies the direction of point creation. Select one of the following:

- **Normal** - Points are created from the start point to the end point.
- **Reversed** - Points are created from the end point to the start point.

Method

Specifies the way points are placed. Select one of the following:

- **By Spacing** - Points are placed at the distance specified in the **Spacing** box.
- **As Divisors** - Points are placed by the value in the **Count** box. The points are spaced equally along the distance between the start and end points.

Spacing

Specifies the distance between each point. This option is only available when **By Spacing** is selected in the **Method** box.

Count

Specifies the number of points to place. This option is only available when **As Divisors** is selected in the **Method** box.

Debug

When set to **On**, displays a sphere around each point to improve visibility. The size of the sphere is set by the value of **Radius**.

- Sphere preview: 
- Sphere after point is created: 

Radius

Specifies the sphere radius for **Debug**.

What do you want to do?

- *Place points along a curve by spacing with start and end points* (on page 1050)
- *Place points along a curve by spacing without start and end points* (on page 1052)
- *Place points along a curve by count with start and end points* (on page 1053)
- *Place points along a curve by count without start and end points* (on page 1054)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place points along a curve by spacing with start and end points

1. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.
2. Create the points needed as start and end points on the curve using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

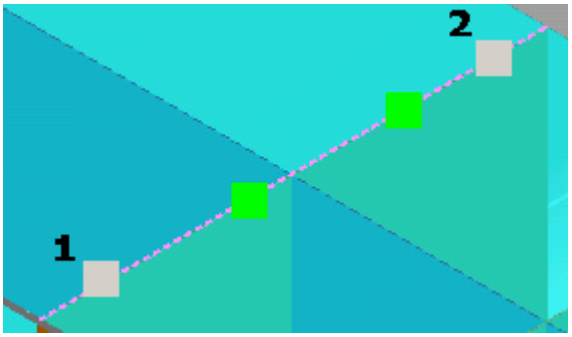
   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
3. Click **PointsAlongCurve** on the **Geometric Construction Palette**.
4. Select the same curve.
5. Click (2), and select a start point for the point pattern.
6. Click (3), and select an end point for the point pattern.

7. In the **Method** box, select **By Spacing**.
8. In the **Direction** box, select **Normal**.
9. In the **Spacing** box, type a value for the distance between points.

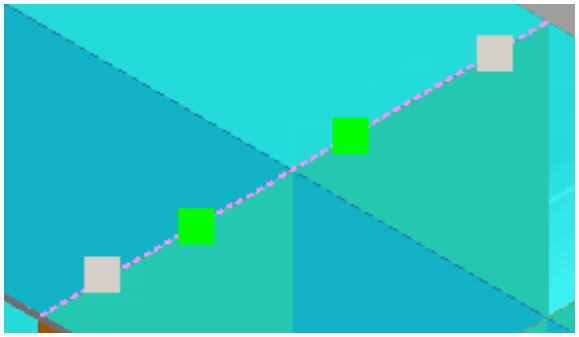
   A preview of the points appears. Spacing starts from the selected start point on the curve and continues until the remaining curve length to the selected end point is less than the spacing value.



***1** - Start point*
***2** - End point*

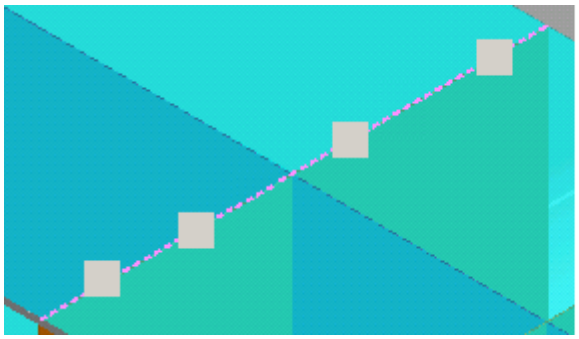
10. If the point spacing does not start from the needed end of the curve, select **Reversed** in the **Direction** box.

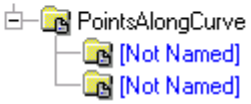
    Points start from the opposite end in the preview.



11. Click **Continue**.

    *Temporary geometry is displayed in the graphic view and as **PointsAlongCurve** > **[Not Named]** in the **Geometric Construction Explorer**.*





12. If needed, create additional geometric constructions.
13. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

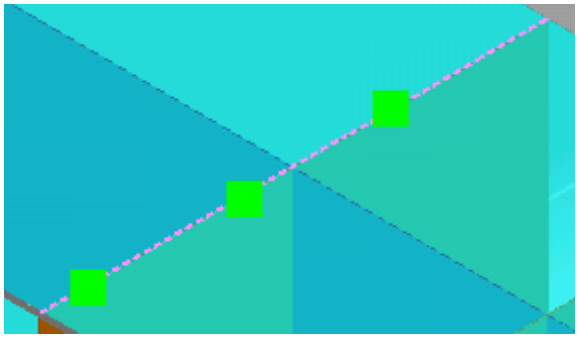
  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place points along a curve by spacing without start and end points

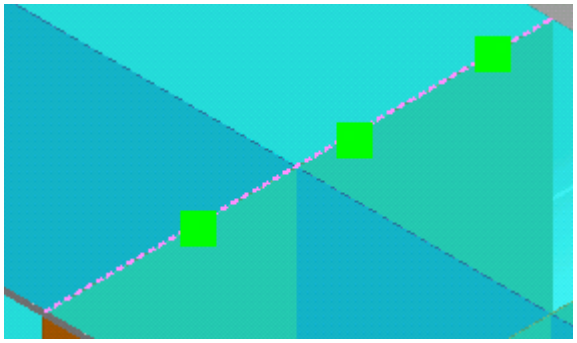
1. Click **PointsAlongCurve** on the **Geometric Construction Palette**.
2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.
3. In the **Method** box, select **By Spacing**.
4. In the **Direction** box, select **Normal**.
5. In the **Spacing** box, type a value for the distance between points.

   A preview of the points appears. Spacing starts from the system-defined start point of the selected curve and continues until the remaining curve length is less than the spacing value.



6. If the point spacing does not start from the needed end of the curve, select **Reversed** in the **Direction** box.

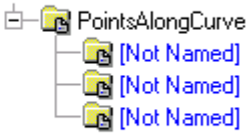
   Points start from the opposite end in the preview.



7. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointsAlongCurve** > **[Not Named]** in the **Geometric Construction Explorer**.*





8. If needed, create additional geometric constructions.
9. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place points along a curve by count with start and end points

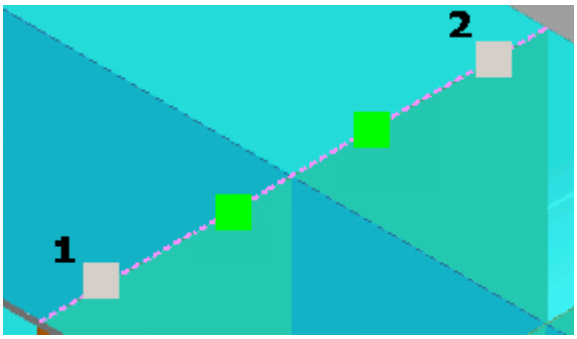
1. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.
2. Create the points needed as start and end points on the curve. These can be points created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
3. Click **PointsAlongCurve** on the **Geometric Construction Palette**.
4. Select the same curve.
5. Click (2), and select a start point for the point pattern.
6. Click (3), and select an end point for the point pattern.
7. In the **Method** box, select **As Divisors**.
8. In the **Count** box, type the number of points to place.

A preview of the points appears. The points are evenly spaced between the selected start and end points.

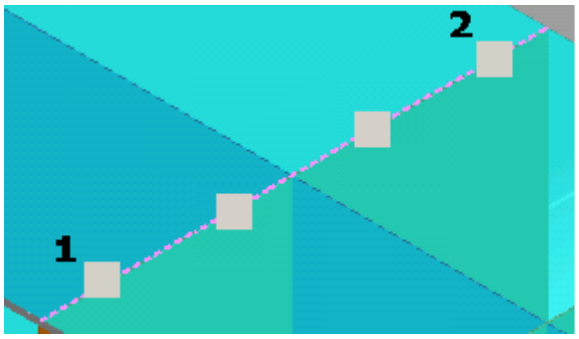


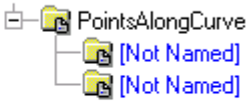
***1** - Start point*

***2** - End point*

9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointsAlongCurve** > **[Not Named]** in the **Geometric Construction Explorer**.*





10. If needed, create additional geometric constructions.
11. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

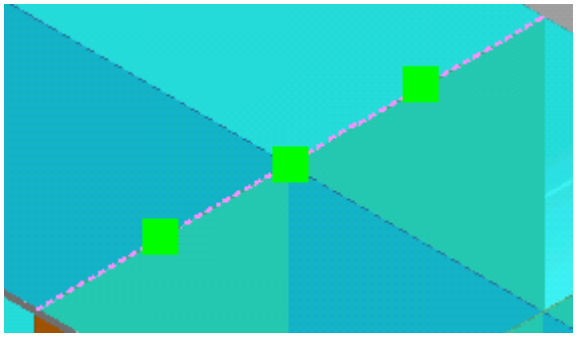
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place points along a curve by count without start and end points

1. Click **PointsAlongCurve** on the **Geometric Construction Palette**.
2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.
3. In the **Method** box, select **As Divisors**.

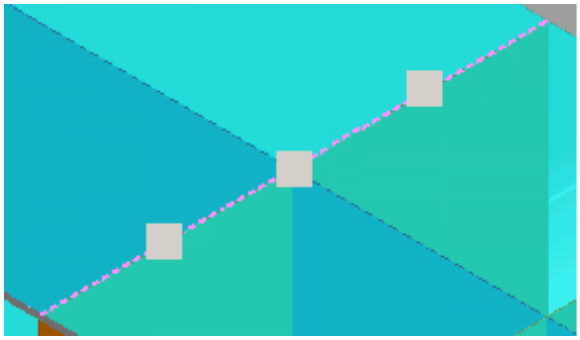
4. In the **Count** box, type the number of points to place.

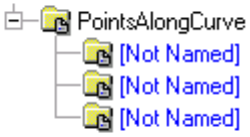
   A preview of the points appears. The points are evenly spaced between the system-defined start and end points of the curve.



5. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointsAlongCurve** > **[Not Named]** in the **Geometric Construction Explorer**.*





6. If needed, create additional geometric constructions.
7. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Modify a geometric construction

Use this procedure to modify a geometric construction that is displayed after clicking **Continue** on a geometric construction ribbon but before clicking **Close** on the geometric construction ribbon.

1. On the geometric construction ribbon, click **Select** .

WARNING Do not click **Select** on the vertical toolbar. This removes you from your current command and deletes all temporary geometric constructions.

2. In a graphic view or the **Geometric Construction Explorer**, select the geometric construction to modify.

   *The geometric construction highlights in the graphic view, the ribbon for the geometric construction appears, and inputs highlight in the **Workspace Explorer**.*

3. Make the required changes.
4. Click **Accept**.

   A preview of the changed geometric construction appears.

5. Click **Continue**.

   Temporary geometry is displayed in the graphic view.

6. If required, create or modify additional geometric constructions.

Change a geometric construction to a different definition

Use this procedure when the geometric construction is displayed after clicking **Continue** on a geometric construction ribbon but before clicking **Close** on the geometric construction ribbon.

1. On the geometric construction ribbon, click **Select**.

   WARNING Do not click **Select** on the vertical toolbar. This removes you from your current command and deletes all temporary geometric constructions.

2. In a graphic view or the **Geometric Construction Explorer**, select the geometric construction to modify.

   *The geometric construction highlights in the graphic view, the ribbon for the geometric construction appears, and the inputs highlight in the **Workspace Explorer**.*

3. In the **Macro Definition** box, click **More**.

   *The **Select Geometric Construction** dialog box appears showing the appropriate geometric constructions. For example, if the original geometric construction is a point, then only points appear.*

4. Select the needed geometric construction, and then click **OK**.

   *The new selection appears in the **Macro Definition** box with a yellow background. Options with blue backgrounds contain new values suggested by the software. Options with white backgrounds retain the values from the previous geometric construction.*

5. Make the required changes to options on the ribbon.

   Changed options appear with a yellow background.

6. Click **Accept**.

   A preview of the changed geometric construction displays.

7. Click **Continue**.

   Temporary geometry is displayed in the graphic view.

8. If required, create or modify additional geometric constructions.

Delete a geometric construction

Use this procedure to delete a geometric construction that is displayed after clicking **Continue** on a geometric construction ribbon but before clicking **Close** on the geometric construction ribbon and then **Finish** on the main command ribbon.

1. On the geometric construction ribbon, click **Select** .

   WARNING Do not click **Select** on the vertical toolbar. This removes you from your current command and deletes all temporary geometric constructions.

2. In a graphic view or the **Geometric Construction Explorer**, select the geometric construction to delete.

   *The geometric construction highlights in the graphic view, the ribbon for the geometric construction appears, and inputs highlight in the **Workspace Explorer**.*

3. On the geometric construction ribbon, click **Delete** .

   The geometric construction is deleted.

4. If required, click **Select** and delete additional geometric constructions.

5. If required, click **Add** to create or modify additional geometric constructions.

Vertices from Curve or Surface



Tooltip: VerticesFromCurveOrSurf

Creates points at the vertices of a curve or surface, including profile and seam landing curves and plate systems. A point is created at each end point, corner point, each change in tangency of 1° or more on a curve, and at each location where a curve crosses a surface patch boundary.

Required Input	Output
Plate part or profile part	Points

Vertices from Curve or Surface Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

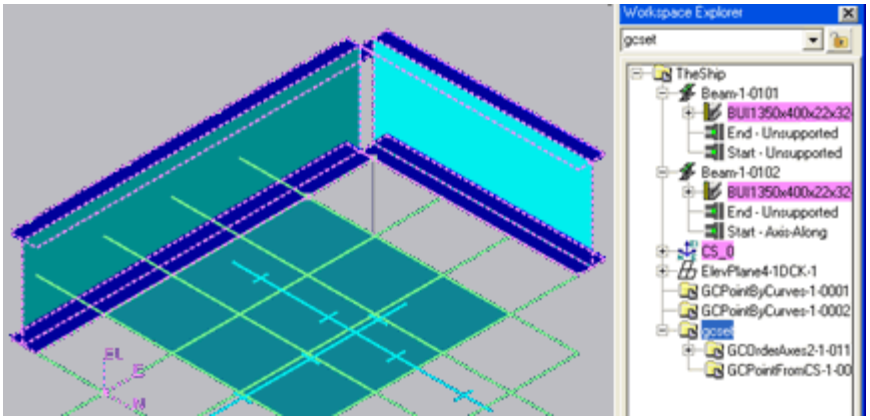
Specifies a surface or curve.

Color Coding

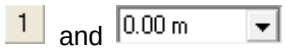
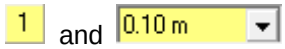
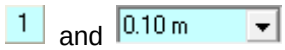
For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

-  - No background color when you keep the value of the original geometric construction.
-  - Yellow background when you change a value.
-  - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

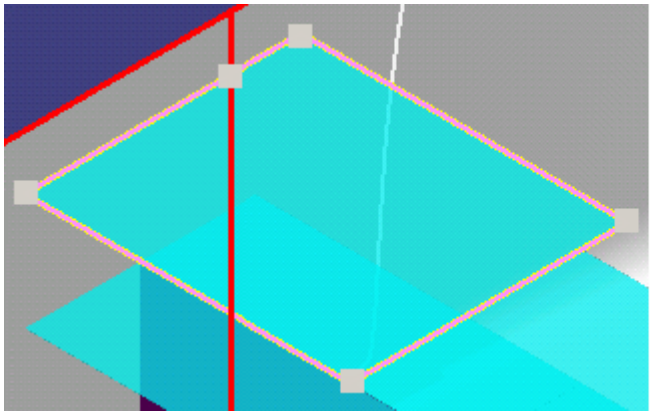
- *Place points on vertices* (on page 1060)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

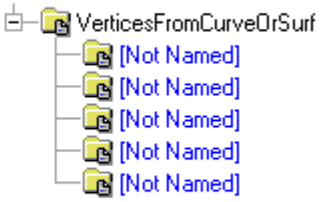
Place points on vertices

1. Click **VerticesFromCurveOrSurf** on the **Geometric Construction Palette**.
2. Select a plate part or profile part.

   Green preview points at all vertices are visible.
3. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **VerticesFromCurveOrSurf** > **[Not Named]** in the **Geometric Construction Explorer**. In the example below, a point is place at each corner and at the surface patch boundary of a bounding plate.*





4. If needed, create additional geometric constructions.
5. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

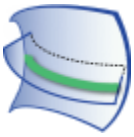
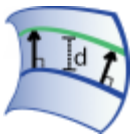
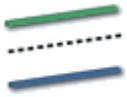
  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

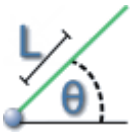
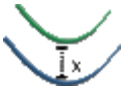
  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Curves (Geometric Construction Palette Dialog Box)

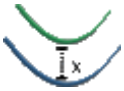
The following geometric construction interfaces create lines and curves.

NOTE Each command supporting the 3D option displays tabs and interfaces appropriate to the command.

	**Curve by Points on Surface** - Creates a curve by interpolation through a set of points. Some points can be designated as knuckles so that the curve has discontinuous tangency. For more information, see *Curve by Points on Surface* (on page 1076).
	**Curve by Two Points on Surface** - Creates a curve between two points projected onto a surface. The curve can be created with or without extensions at each end. For more information, see *Curve by Two Points on Surface* (on page 1081).
	**Fillet by Curve** - Creates a blended fillet of a specified radius between two curves that are end-connected and projected onto a surface. For more information, see *Fillet by Curve* (on page 1090).
	**Curve by Curves** - Creates a single curve from two or more end-to-end curves. For more information, see *Curve by Curves* (on page 1070).
	**Curve by Intersection** - Creates a curve along the intersection of two surfaces. For more information, see *Curve by Intersection* (on page 1073).
	**Curve at Offset by Intersection** - Creates a curve offset from the intersection of a surface or plane with a surface. For more information, see *Curve at Offset by Intersection* (on page 1066).
	**Line from Coordinate System** - Creates a line along a coordinate system axis. For more information, see *Line from Coordinate System* (on page 1100).
	**Line by Points** - Creates a line between two points. For more information, see *Line by Points* (on page 1098).
	**Parallel Line** - Creates a line parallel to another line. For more information, see *Parallel Line* (on page 1107).

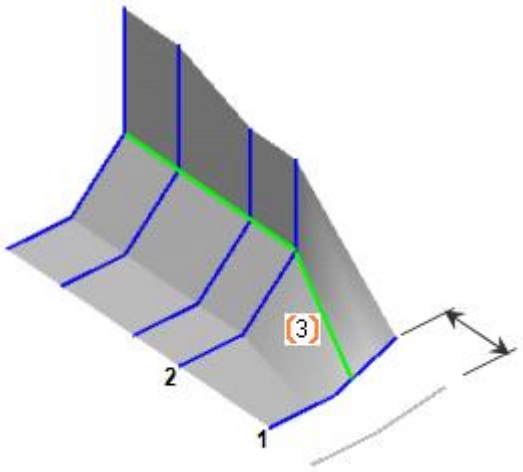
	**Line by Point, Angle, and Length** - Creates a line at a defined angle and length on a coordinate system plane. For more information, see *Line by Point, Angle, and Length* (on page 1094).
	**Line String by Points** - Creates a line string between a series of two or more points. For more information, see *Line String by Points* (on page 1104).
	**Edge from Curve or Surface** - Creates a curve at an edge of a surface, such as a plate, the landing curve of a profile, or a solid. For more information, see *Edge from Curve or Surface* (on page 1085).
	**Edges from Curve or Surface** - Creates one or more curves at each edge of a surface, such as a plate, the landing curve of a profile, or a solid. For more information, see *Edges from Curve or Surface* (on page 1088).
	**Polyline by One to Four Segments** - Creates a polyline string of up to four curves by defining point coordinates on a reference plane. For more information, see *Polyline by One to Four Segments* (on page 1110).
	**Curve as Extended Section** - Copies polyline cross-sections. For more information, see *Curve as Extended Section* (on page 1063).

Curve as Extended Section



Tooltip: CurveAsExtendedSection

Copies cross-sections created using the **Polyline by One to Four Segments** (on page 1110) geometric construction. This curve type is commonly used to extend a surface created with the **Surface by Curves** (on page 1130) geometric construction. You can add a cross-section before the first cross-section and after the last cross-section of the surface.



Required Input	Output
1 - Polyline to extend	Extended polyline
2 - Second polyline	
Optional Input	
(3) - Line string joining equivalent vertices	
Parameters	
Extension distance	

Curve as Extended Section Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first polyline cross-section. This is typically a cross-section at the end of a surface created with the **Surface by Curves** (on page 1130) geometric construction.

2 - Specifies the second polyline cross-section. This is typically the cross-section next to the selected end of the surface.

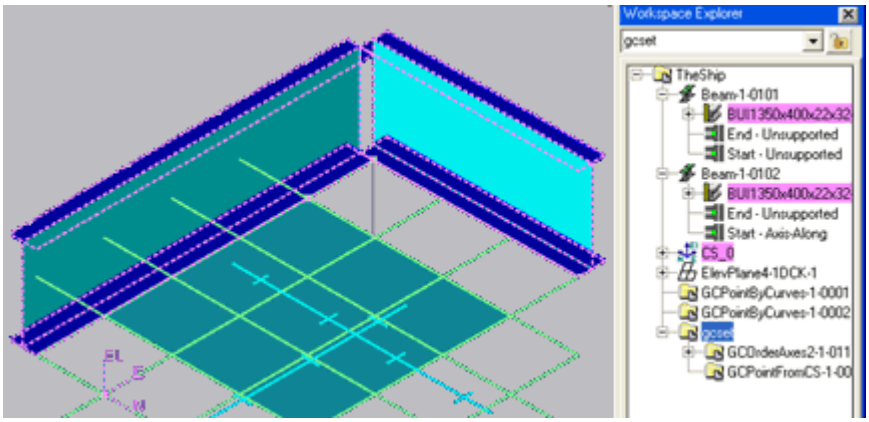
(3) - Specifies a line string joining equivalent vertices of the cross-sections. This value is optional, and is only needed when the cross-sections do not have the same number of vertices.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Cancel - Clears the current selections.

Accept - Accepts the current selections, and displays a preview.

Continue - Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Extension - Specifies the distance by which the first cross-section **1** is extended.

What do you want to do?

- *Extend a polyline cross-section* (on page 1065)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Extend a polyline cross-section

1. Click **CurveAsExtendedSection** on the **Geometric Construction Palette**.
2. Select the first cross-section.
3. Select the second cross-section.
4. If the cross-sections have different numbers of vertices, click (3), and select a line string joining equivalent vertices.

5. Click **Accept** .

   A preview of the extended cross-section appears.

6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CurveAsExtendedSection** in the **Geometric Construction Explorer**.*

7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

★ IMPORTANT

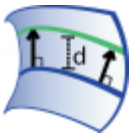
- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Curve at Offset by Intersection



Tooltip: CurveAtOffsetByInt

Creates a curve on a surface offset from an intersecting reference plane or surface.

Required Input	**Output**
Intersecting reference plane or surface	Curve
Intersected surface	
Parameters	
Offset distance	

Curve at Offset by Intersection Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies an intersecting surface or reference plane.

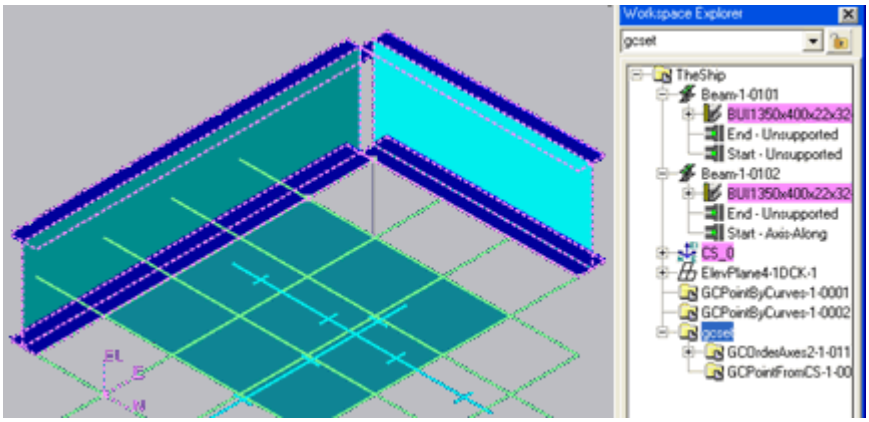
2 - Specifies the intersected surface or reference plane. By default, this is the surface of the plate selected for the main command.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Offset - Defines the offset distance of the curve from the intersecting reference plane or surface.

What do you want to do?

- *Place a curve offset from an intersection* (on page 1069)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a curve offset from an intersection

1. Click **CurveAtOffsetByInt** on the **Geometric Construction Palette**.
2. Select a surface, such as a plate system or a reference plane. Typically this is a surface intersecting the surface selected for the main command.
3. Select a surface intersected by the first surface. By default, this is the surface selected for the main command.
4. Type an offset value.

   A green preview curve is visible at the offset from the surface intersection. If one of the surfaces, such as a reference plane, is infinite, then the preview curve is infinite.
5. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CurveAtOffsetByInt** in the **Geometric Construction Explorer**.*
6. If needed, create additional geometric constructions.
7. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Curve by Curves



Tooltip: CurveByCurves

Creates a single curve from two or more end-to-end curves.

Required Input	Output
Two or more connected curves	Curve

Curve by Curves Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

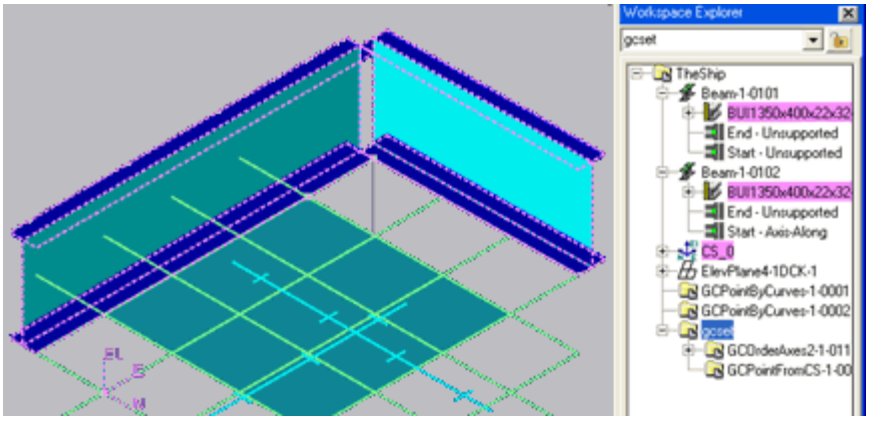
1 - Specifies two or more connected curves to combine.

Color Coding

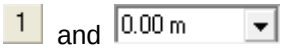
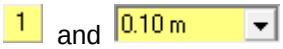
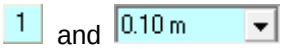
For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background color when there is no input.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

-  - No background color when you keep the value of the original geometric construction.
-  - Yellow background when you change a value.
-  - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Combine end-to-end curves* (on page 1072)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Combine end-to-end curves

1. Create the curves needed to define the fillet. These curves must be connected and are created with other geometric construction curve interfaces.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)
2. Click **CurveByCurves** on the **Geometric Construction Palette**.
3. Select two or more end-to-end curves. Select the curves in sequence.
4. Click **Continue**.

   The curves are merged into a single curve.
5. If needed, create additional geometric constructions.
6. Click **Close** to return to the main command ribbon.

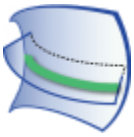
IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Curve by Intersection



Tooltip: CurveByIntersection

Creates a curve along the intersection of two surfaces.

Required Input	**Output**
Intersecting surface	Curve
Intersected surface	
Optional Input	
Coordinate system	

Curve by Intersection Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult

your Intergraph support representative before using these geometric constructions.

1 - Specifies an intersecting surface or reference plane.

2 - Specifies the intersected surface or reference plane. By default, this is the surface of the plate selected for the main command.

(3) - Specifies a coordinate system used to orient the curve. This value is optional and is not available when **Keep all curves** is selected for **Curve Option**.

(4) - Specifies a tracking point used to control the direction of curve placement when an ambiguity exists. A tracking point is only needed with curved surfaces that intersect more than once. The tracking point is used with the value selected by **Track Flag**. This value is optional and is not available when **Keep all curves** is selected for **Curve Option**.

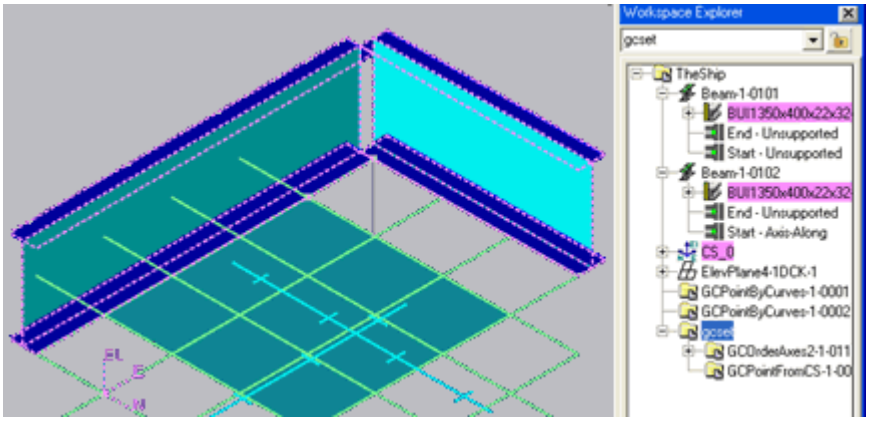
NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Track Flag - Controls how the tracking point is used when the **Curve Option** of **Keep one curve** is selected and an ambiguity exists. Select **Near** to use the curve nearest to the tracking point. Select **Far** to use the curve farthest from the tracking point.

Curve Option - Controls the number of curves that are created when an ambiguity exists. Select **Keep one curve** to create one curve based on the selected value of **Track Flag** and the selected tracking point. Select **Keep all curves** to create all possible curves.

What do you want to do?

- *Place a curve by intersection* (on page 1075)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a curve by intersection

1. Click **CurveByIntersection** on the **Geometric Construction Palette**.
2. Select a surface, such as a plate system or a reference plane. Typically this is a surface intersecting the surface selected for the main command.
3. Select a surface intersected by the first surface. By default, this is the surface selected for the main command.

   A green preview curve is visible at the surface intersection. If one of the surfaces, such as a reference plane, is infinite, then the preview curve is infinite.
4. Optionally, click (3), and select a coordinate system to orient the curve.
5. To create all possible curves, select **Keep all curves** for **Curve Option**.
6. To create one curve when multiple curves are possible:
   - Select **Keep one curve** for **Curve Option**.
   - Click (4), and select the needed value for **Track Flag**.

   NOTE If changing the value for **Track Flag** does not give the needed solution, you can first create a point that you then select after clicking (4). The point can be created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces:

Control Point in the *Common User's Guide*

Topological Points (Insert Menu) (on page 1323)

Points (Geometric Construction Palette Dialog Box) (on page 1028)

7. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CurveByIntersection** in the **Geometric Construction Explorer**.*

8. If needed, create additional geometric constructions.
9. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Curve by Points on Surface



Tooltip: CurveByPointsOnSurface

Creates a curve by interpolation through a set of points. Some points can be designated as knuckles so that the curve has discontinuous tangency.

Required Input	**Output**
Points	Curve
Surface	
Optional Input	
Knuckle points	
Tangent vectors at start and end	

Curve by Points on Surface Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the points used for the curve.

★IMPORTANT Select the points in the order in which the curve is created.

(2) - Specifies points selected in **1** to use as knuckles. This value is optional.

(3) - Specifies a line or edge as a start vector used to control the interpolation. The vector acts as a tangent at the start point of the curve. This value is optional.

(4) - Specifies a line or edge as an end vector used to control the interpolation. The vector acts as a tangent at the end point of the curve. This value is optional.

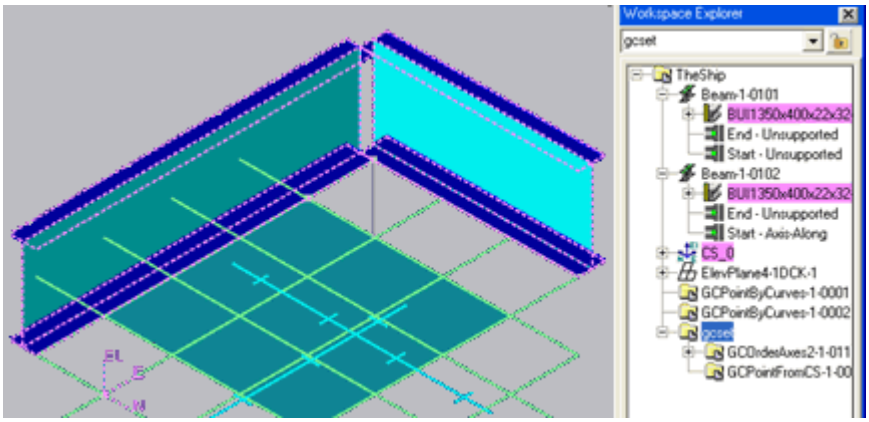
5 - Specifies the surface for the curve.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a curve by points* (on page 1079)
- *Place a knuckled curve by points* (on page 1080)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a curve by points

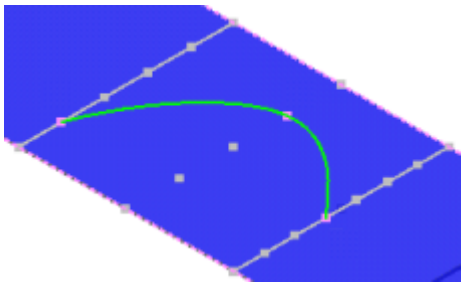
1. Create the points needed to define the curve by using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

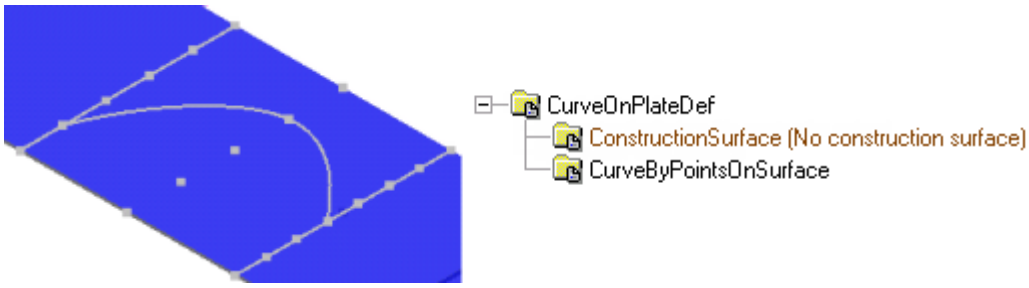
   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **CurveByPointsOnSurface** on the **Geometric Construction Palette**.
3. Select the points used to define the curve. Select the points in their order on the curve.
4. Optionally, click (3), and select a start vector.
5. Optionally, click (4), and select an end vector.
6. Click **5**, and select the surface for the curve.

   A preview of the curve appears. The curve passes through the points in the selected order.



7. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CurveByPointsOnSurface** in the **Geometric Construction Explorer**.*



8. If needed, create additional geometric constructions.
9. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place a knuckled curve by points

1. Create the points needed to define the curve. These can be points created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

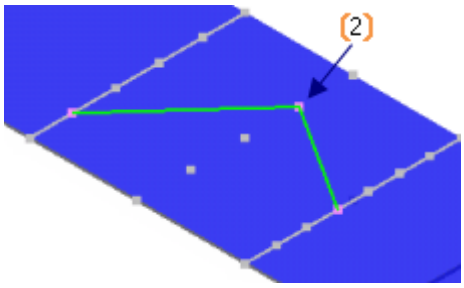
   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)

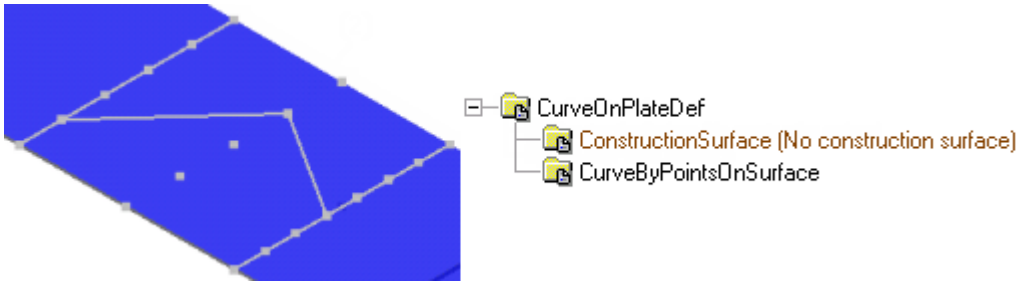
2. Click **CurveByPointsOnSurface** on the **Geometric Construction Palette**.
3. Select the points used to define the curve. Select the points in their order on the curve.
4. Click (2), and select one or more of the previously-selected points (excluding the end points) to use as knuckles.
5. Optionally, click (3), and select a start vector.
6. Optionally, click (4), and select an end vector.
7. Click **5**, and select the surface for the curve.

   A preview of the curve appears. The curve passes through the points in the selected order and is knuckled at the selected points. One knuckle point is shown in the example below.



8. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CurveByPointsOnSurface** in the **Geometric Construction Explorer**.*



9. If needed, create additional geometric constructions.

10. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Curve by Two Points on Surface



Tooltip: CurveBy2PointsOnSurface

Creates a curve between two points projected onto a surface along the surface normal, an axis, or angle. The curve can be created with or without extensions at each end.

Required Input	**Output**
Start and end points	Curve
Surface	
Coordinate system	
Parameters	
Projection method	
Start and end projections	

Curve by Two Points on Surface Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first point.

2 - Specifies the second point.

3 - Specifies the surface onto which the curve is projected. By default, this is the surface of the plate selected for the main command.

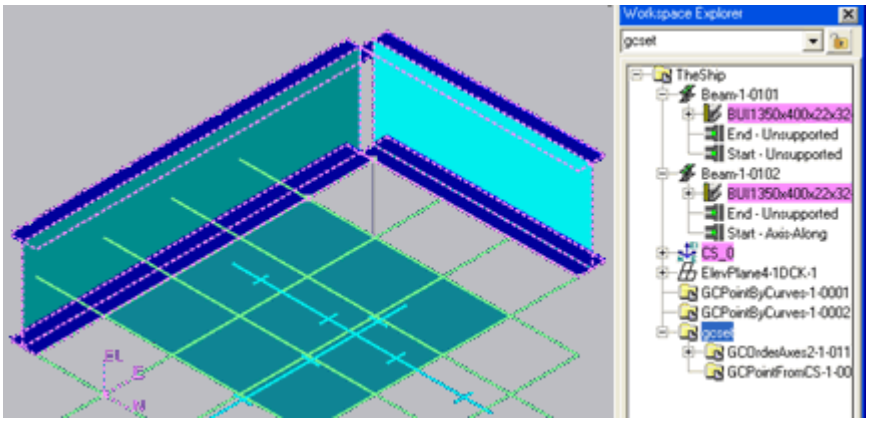
4 - Specifies the coordinate system used to control the direction of projection.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Projection Method - Specifies how the curve is projected to the surface. Select one of the following:

- **Normal** - Projects the curve normal to the surface.
- **Along x** - Projects the curve along the x-axis of the selected coordinate system.
- **Along y** - Projects the curve along the y-axis of the selected coordinate system.
- **Along z** - Projects the curve along the z-axis of the selected coordinate system.
- **At Angle** - Projects the curve at an angle.

Angle - Specifies the projection angle relative to the reference axis. This option is only available when **Angle** is selected for **Projection Method**.

Rotation Axis - Specifies the rotation axis for an angle projection. Select **x**, **y**, or **z**. This option is only available when **Angle** is selected for **Projection Method**.

Reference Axis - Specifies the reference axis for an angle projection. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is only available when **Angle** is selected for **Projection Method**.

Extension at Start - Specifies the distance that the curve is extended before the first point.

Extension at End - Specifies the distance that the curve is extended after the second point.

What do you want to do?

- *Place a curve on a surface by two points and projection* (on page 1084)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a curve on a surface by two points and projection

1. Create the points needed to define the curve. These can be points created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **CurveBy2PointsOnSurface** on the **Geometric Construction Palette**.
3. Select the first point for the curve.
4. Select the second point for the curve.
5. Select the surface. By default, this is the surface of the plate selected for the main command.
6. Select a coordinate system to use for projection of the curve.
7. Select a projection method. If **Angle** is selected, also select the needed values for **Rotation Axis** and **Reference Axis**.
8. Optionally, type values for extensions at the start and end of the curve.

   A preview of the curve appears.
9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CurveBy2PointsOnSurface** in the **Geometric Construction Explorer**.*
10. If needed, create additional geometric constructions.
11. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

"The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Edge from Curve or Surface



Tooltip: EdgeFromCurveOrSurf

Creates a curve at an edge of a surface, such as a plate, the landing curve of a profile, or a solid.

Required Input	Output
Curve, surface, or solid	Curve

Edge from Curve or Surface Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies a surface, such as a plate, the landing curve of a profile or seam, or a solid.

(2) - Specifies a tracking point used to control curve placement on an edge when an ambiguity exists. A tracking point is only needed with curved surfaces that intersect more than once. The tracking point is used with the value selected by **Track Flag**. This value is optional.

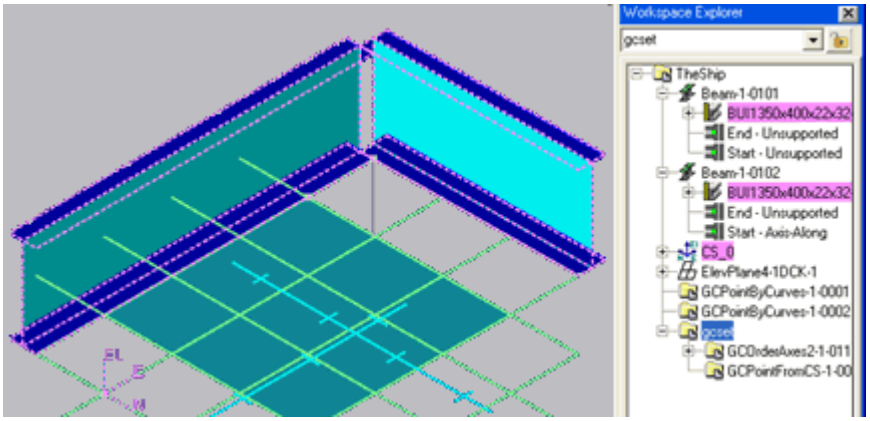
NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Track Flag - Controls how the tracking point is used when an ambiguity exists. Select **Near** to use the curve nearest to the tracking point. Select **Far** to use the curve farthest from the tracking point.

What do you want to do?

- *Place a curve at an edge* (on page 1087)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a curve at an edge

1. Click **EdgeFromCurveOrSurf** on the **Geometric Construction Palette**.
2. Select a plate or profile system or part, a seam, or a solid.

   A green preview curve is visible on an edge.
3. If there are multiple solutions, such as on the edges of a plate, click (2), select a tracking point, and select the needed value for **Track Flag**.

   NOTE If changing the value for **Track Flag** still does not give the needed solution, you may need to first create a point that you then select after clicking (4). The point can be created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
4. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **EdgeFromCurveOrSurf** in the **Geometric Construction Explorer**.*
5. If needed, create additional geometric constructions.
6. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Edges from Curve or Surface



Tooltip: EdgesFromCurveOrSurf

Creates a curve at each edge of a surface, such as a plate or the landing curve edge of a profile.

Required Input	Output
Curve, surface, or solid	Curve on each edge

Edges from Curve or Surface Ribbon

 **Select**

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

 **Add**

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

 **Delete**

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

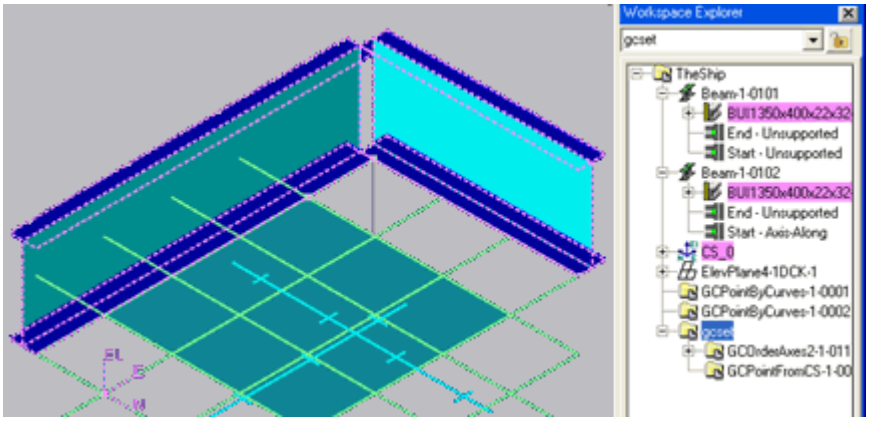
1 - Specifies a surface, such as a plate, the landing curve of a profile or seam, or a solid.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a curve on all edges of a selected object* (on page 1090)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a curve on all edges of a selected object

1. Click **EdgesFromCurveOrSurf** on the **Geometric Construction Palette**.
2. Select a plate or profile system or part, a seam, or a solid.

   A green preview curve is visible on each edge.
3. Click **Continue**.

   *Temporary geometry is displayed in the graphic view. The curve at each edge also appears as **EdgesFromCurveOrSurf** > **[Not Named]** in the **Geometric Construction Explorer**.*
4. If needed, create additional geometric constructions.
5. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Fillet by Curve



Tooltip: CmdFilletByCurve

Creates a blended fillet of a specified radius between two curves that are end-connected and projected onto a surface.

Required Input	**Output**
Two end-connected curves Surface **Optional Input** Projection Method Radius	Curve with radius and tangential to input curves

Fillet by Curve Ribbon

 **Select**

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

 **Add**

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

 **Delete**

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click  again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first curve.

2 - Specifies the second curve. This curve must be connected to the first curve.

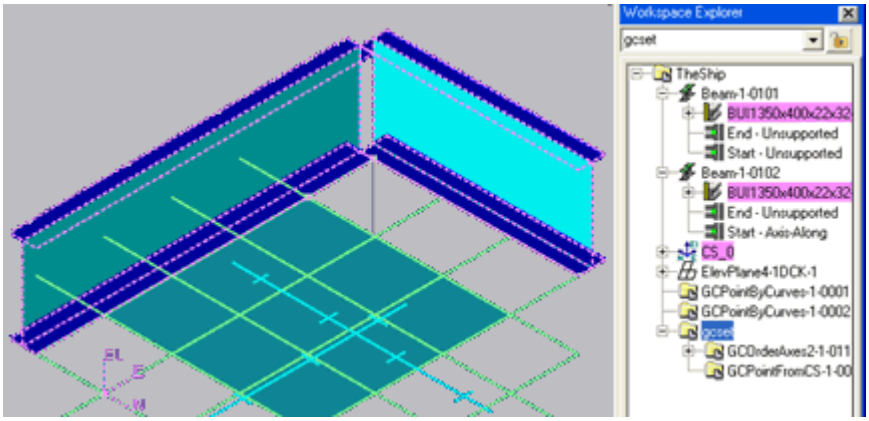
3 - Specifies the surface onto which the curves and the fillet are projected. By default, this is the surface of the plate selected for the main command.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Radius - Specifies the radius of the fillet curve.

Projection Method - Specifies how the curve is projected to the surface. Select one of the following:

- **Normal** - Projects the curve normal to the surface.
- **Along x** - Projects the curve along the x-axis of the selected coordinate system.
- **Along y** - Projects the curve along the y-axis of the selected coordinate system.
- **Along z** - Projects the curve along the z-axis of the selected coordinate system.

What do you want to do?

- *Place a filleted curve on a surface* (on page 1093)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a filleted curve on a surface

1. Create the curves needed to define the fillet. These curves must be connected and are created with other geometric construction curve interfaces.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)
2. Click **CmdFilletByCurve** on the **Geometric Construction Palette**.
3. Select the first curve.
4. Select the second curve.
5. Select the surface. By default, this is the surface of the plate selected for the main command.
6. Type a radius for the fillet.
7. Select a projection method.

   A preview of the projected curves and fillet appears with the first and second curves tangential to and trimmed to the fillet.
8. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CmdFilletByCurve** in the **Geometric Construction Explorer**.*
9. If needed, create additional geometric constructions.
10. Click **Close** to return to the main command ribbon.

★ IMPORTANT

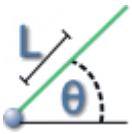
- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

"The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Line by Point, Angle, and Length



Tooltip: LineByPointAngleLength

Creates a line at a defined angle and length on a coordinate system plane.

Required Input	**Output**
Coordinate system	Line
Starting point	
Optional Input	
Reference line	
Parameters	
Coordinate system plane	
Length	
Angle	

Line by Point, Angle, and Length Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction

ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies a coordinate system used to orient the line and define the plane of the line.

2 - Specifies the starting point of the line.

(3) - Specifies a reference line. The angle of the line is measured from the reference line. If no reference line is selected, the angle is based on the coordinate system axes. This value is optional.

(4) - Specifies a tracking point used to control the direction of curve placement when an ambiguity exists. The tracking point is used with the value selected by **Track Flag**. This value is optional.

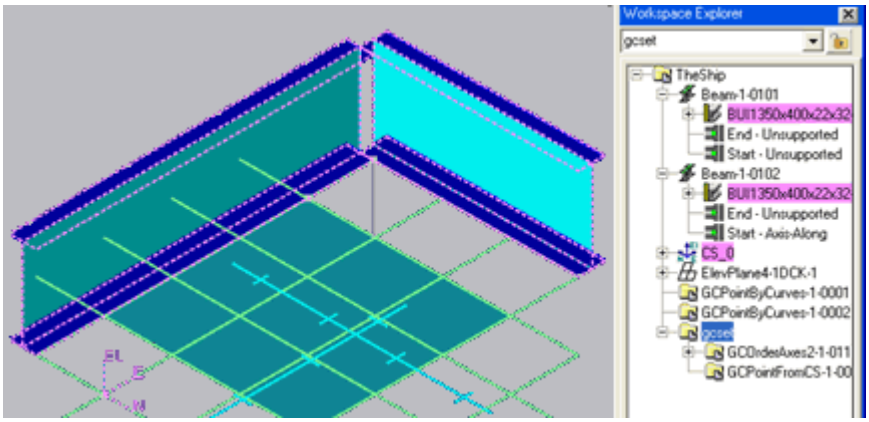
NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Active Axes - Defines the coordinate system plane on which the line is placed. Select **(x,y)**, **(y,z)**, **(z,x)**, **(y,x)**, **(z,y)**, or **(x, z)**.

Angle - Defines the angle of the line about the axis normal to the plane defined by **Active Axes**. If a reference line is selected for (3), then the angle is measured from that line.

Length - Defines the length of the line.

Track Flag - Controls how the tracking point (4) is used when an ambiguity exists. Select **Near** to use the curve nearest to the tracking point. Select **Far** to use the curve farthest from the tracking point.

What do you want to do?

- *Place a line by point, angle, and length* (on page 1097)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a line by point, angle, and length

1. Create the point needed as the start point of the line using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. If you are using a reference line, create that line with another geometric construction line interface.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)
3. Click **LineByPointAngleLength** on the **Geometric Construction Palette**.
4. Select the coordinate system.
5. Select the starting point.
6. Optionally, click (3), and select a reference line.
7. In the **Active Axes** box, select the plane for the line.
8. Type a value for **Angle**.
9. Type a value for **Length**.

   A preview of the line appears.
10. If there are multiple solutions, such as the edges of a plate, click (4), select a tracking point, and select the needed value for **Track Flag**.
11. Click **Continue**.

    *Temporary geometry is displayed in the graphic view and as **LineByPointAngleLength** in the **Geometric Construction Explorer**.*
12. If needed, create additional geometric constructions.
13. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Line by Points



Tooltip: LineByPoints

Creates a line between two points.

Required Input	Output
Starting and ending point	Line

Line by Points Ribbon

 **Select**

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

 **Add**

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

 Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click  again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first point.

2 - Specifies the second point.

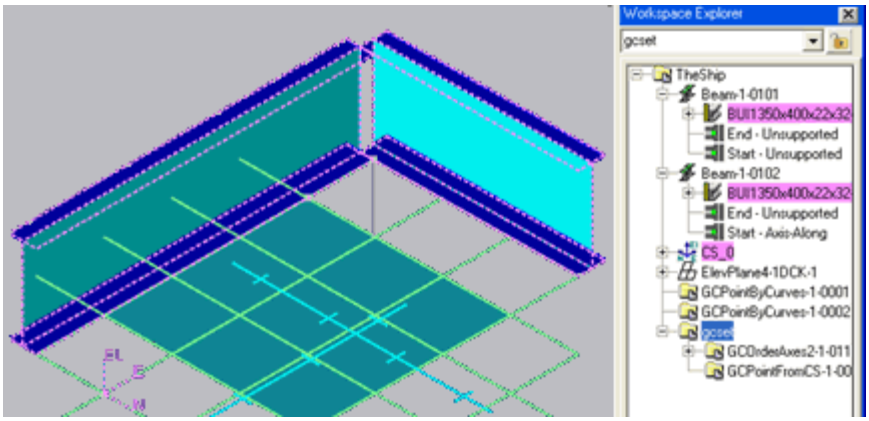
NOTE If successive lines are created using **Line by Points**, the point selected for **2** of a line is the default selection for **1** of the next line.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a line between two points* (on page 1100)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a line between two points

1. Create the points needed as the start and end points of the line using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **LineByPoints** on the **Geometric Construction Palette**.
3. Select the first point.
4. Select the second point.

   A preview of the line appears.
5. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **LineByPoints** in the **Geometric Construction Explorer**.*
6. If needed, create additional geometric constructions.

   NOTE If successive lines are created using **Line by Points**, the point selected for **2** of a line is the default selection for **1** of the next line.
7. Click **Close** to return to the main command ribbon.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon and create the model object.

Line from Coordinate System



Tooltip: LineFromCS

Creates a line along a coordinate system axis.

Required Input	Output
Coordinate system	Line
Optional Input	
Starting point	
Parameters	
Looking axis	
Length, orientation, and justification	

Line from Coordinate System Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies a coordinate system used to orient the line.

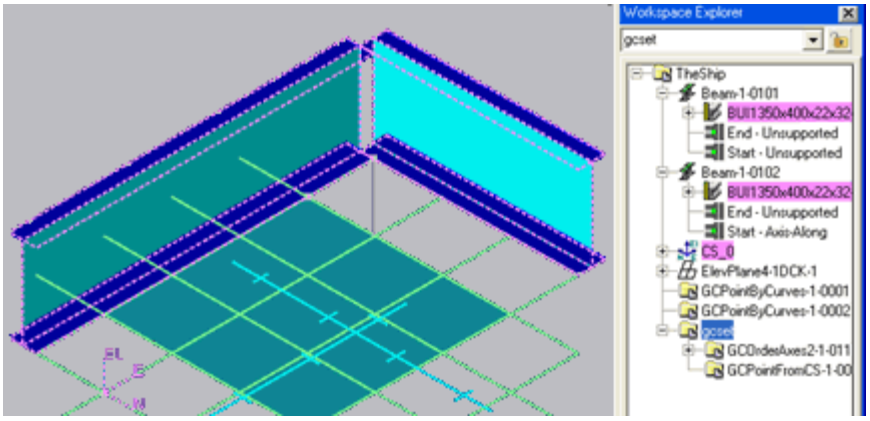
(2) - Specifies a starting point for the line. If no starting point is selected, the origin of the coordinate system is used. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Looking Axis - Specifies the coordinate system axis of the line. The line is place in the direction of the axis. Select **X**, **Y**, or **Z**.

Length - Defines the length of the line.

Orientation - Specifies the direction of the line from the starting point. Select **Direct** to place the line in the positive axis direction. Select **Indirect** to place the line in the negative axis direction.

NOTE Orientation is ignored if **Centered** is selected for **Justification**.

Justification - Specifies the placement of the line relative to the starting point. Select **Centered** to center the line on the starting point. Select **Not Centered** to place the line according the value selected for **Orientation**.

What do you want to do?

- *Place a line in the direction of a coordinate system axis* (on page 1103)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a line in the direction of a coordinate system axis

1. Create the point needed as the start point of the line using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **LineFromCS** on the **Geometric Construction Palette**.
3. Select the coordinate system.
4. Optionally, click (2), and select a starting point.
5. In the **Looking Axis** box, select an axis.
6. Type a value for **Length**.
7. Select values for **Orientation** and **Justification**.

   A preview of the line appears.
8. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **LineFromCS** in the **Geometric Construction Explorer**.*
9. If needed, create additional geometric constructions.
10. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Line String by Points



Tooltip: LineStringByPoints

Creates an open or closed line string between a series of two or more points.

Required Input	**Output**
Points	Line
Parameter	
Closed or open string	

Line String by Points Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

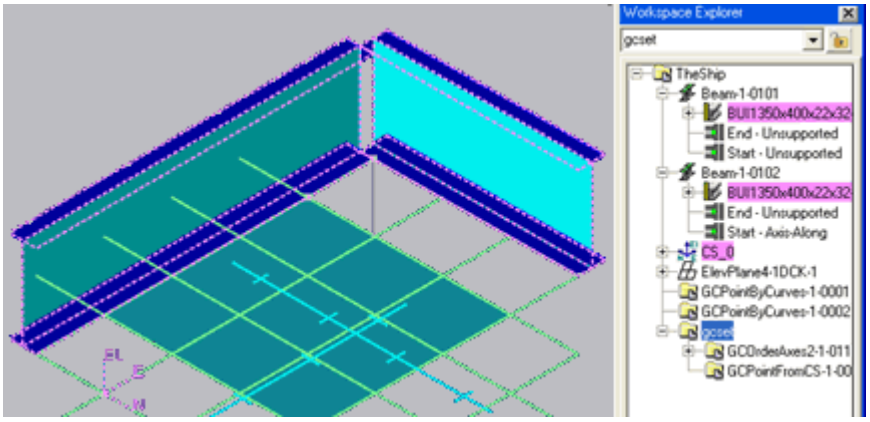
1 - Specifies the points used for the line string.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Closed Flag - Specifies whether the line string is extended from the last point back to the first point, creating a closed string. Select **Closed** to extend the string. Select **Opened** to stop the string at the last point.

What do you want to do?

- *Place a line string* (on page 1106)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a line string

1. Create the points needed to define the line by using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **LineStringByPoints** on the **Geometric Construction Palette**.
3. Select the points used to define the line. Select the points in their order on the line string.
4. To extend the line string from the last point back to the first point, select **Closed** in the **Closed Flag** box.

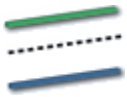
   A preview of the line appears.
5. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **LineStringByPoints** in the **Geometric Construction Explorer**.*
6. If needed, create additional geometric constructions.

   NOTE If successive lines are created using **Line by Points**, the point selected for **2** of a line is the default selection for **1** of the next line.
7. Click **Close** to return to the main command ribbon.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon and create the model object.

Parallel Line



Tooltip: LineParallel

Creates a line parallel to another line.

NOTE This geometric construction only works with lines, not line strings or curves.

Required Input	Output
Line	Line
Surface or plane	
Parameters	
Distance	

Parallel Line Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric**

Construction dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies a line. You must select a line or linear landing curve. You cannot select a line string or a curve.

2 - Select a surface or sketching plane.

(3) - Specifies a tracking point used to control the direction of parallel line placement when an ambiguity exists. The tracking point is used with the value selected by **Track Flag**. This value is optional.

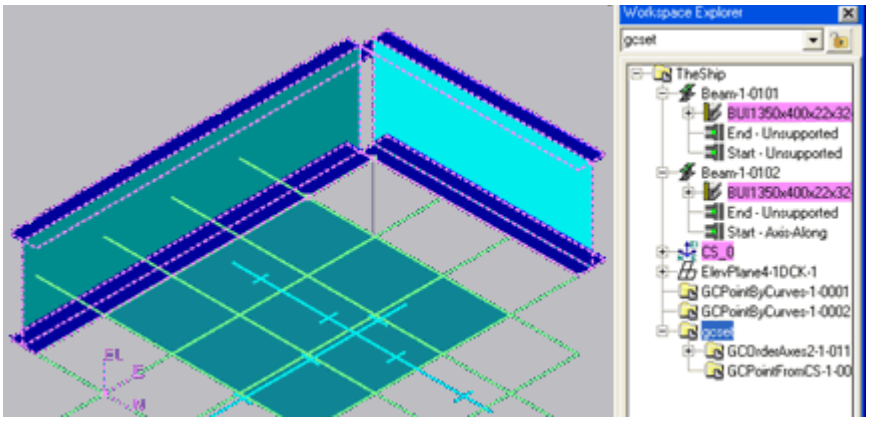
NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Distance - Defines the distance of the parallel line from the selected line.

Track Flag - Controls how the tracking point is used when an ambiguity exists. Select **Near** to use the curve nearest to the tracking point. Select **Far** to use the curve farthest from the tracking point.

What do you want to do?

- *Place a line parallel to another line* (on page 1109)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a line parallel to another line

1. Click **LineParallel** on the **Geometric Construction Palette**.
2. Select a line.
3. Select a sketching plane or surface.
4. Type a value for **Distance**.

   A preview of the parallel line displays.
5. If there are multiple solutions, such as the edges of a plate, click (3), select a tracking point, and select the needed value for **Track Flag**.
6. Click **Continue**.

   *Temporary geometry displays in the graphic view and as **LineParallel** in the **Geometric Construction Explorer**.*
7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

"The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and is unavailable when editing an object.

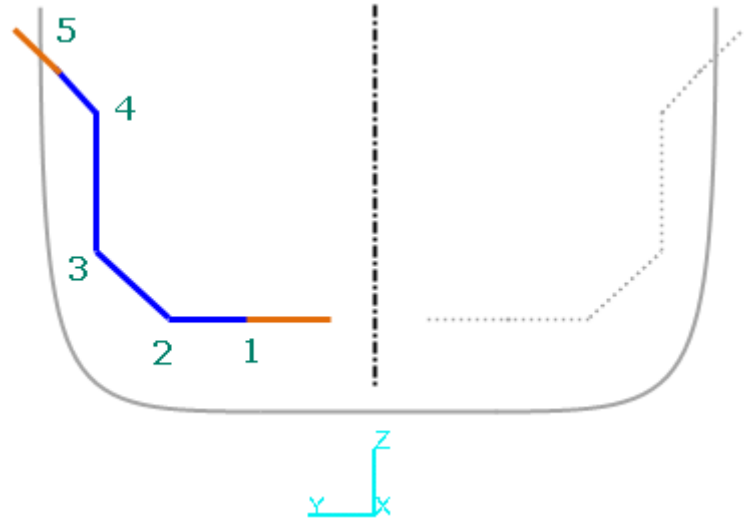
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Polyline by One to Four Segments



Tooltip: PolylineBy1to4Segments

Creates a polyline string of up to four lines by defining point coordinates on a reference plane. This curve type is commonly used to define hopper sides in a bulk carrier merchant ship. Hoppers run longitudinally, so symmetry is defined about the longitudinal center line as +Y side, -Y side, or both sides.



Required Input	**Output**
Reference plane	Polyline
Parameters	
Number of polyline segments (maximum four segments)	
Point coordinates (maximum five points)	
Symmetry	
Start and end extensions	

Polyline by One to Four Segments Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the reference plane for the polyline.

2 - Specifies a reference plane or object along the selected reference axis (**Y reference** or **Z reference**) for **Point #1 Coord**. This option is only available when **Y reference** or **Z reference** is selected for **Point #1 Coord**.

3 - Specifies a reference plane or object normal to the selected axis for **Point #1 Coord**. For example, if **Y axis** or **Y reference** is selected, then the reference plane or object must be in the Z-axis direction. This option is only available when **Reference** is selected for **Point #1 Orient**.

4 - Specifies a reference plane or object along the selected reference axis (**Y reference** or **Z reference**) for **Point #2 Coord**. This option is only available when **Y reference** or **Z reference** is selected for **Point #2 Coord**.

5 - Specifies a reference plane or object normal to the selected axis for **Point #2 Coord** For example, if **Y axis** or **Y reference** is selected, then the reference plane or object must be in the Z-axis direction. This option is only available when **Reference** is selected for **Point #2 Orient**.

6 - Specifies a parallel line for Point #2. This option is only available when **Parallel** is selected for **Point #2 Orient**.

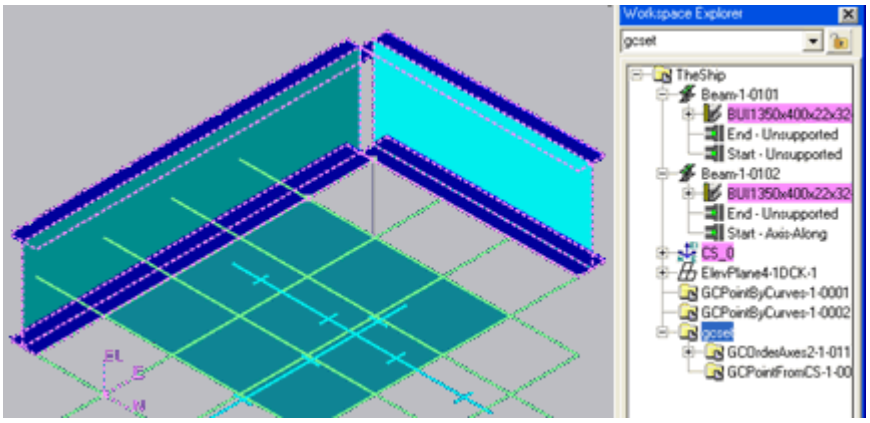
NOTE Options similar to **4**, **5**, and **6** are also available for **Point #3**, **Point #4**, and **Point #5**.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Cancel - Clears the current selections.

Accept - Accepts the current selections, and displays a preview.

Continue - Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Auto Preview - Specifies whether a preview of the polyline is displayed. The preview updates as options are changed. Select **On** or **Off**.

Segments Count - Specifies the number of line segments in the polyline. Select **1**, **2**, **3**, or **4**.

Category - Specifies the type of polyline parameters. Select **Ref Plane**, **Point #1**, **Point #2**, **Point #3**, **Point #4**, **Point #5**, **Othe**r, or **All**. Descriptions of the parameters for each category are below. **All** displays the properties for all categories.

Ref Plane Category

Defines the reference plane offset.

Ref Plane Offset - Type the offset value.

Point #1 Category

Defines the first point on the polyline.

Point #1 Coord - Select **Y axis**, **Z axis**, **Y reference**, or **Z reference**.

- **Point #1 Coord1** - When **Y axis** or **Z axis** is selected for **Point #1 Coord**, type a coordinate value. For **Y axis**, this value is the Y coordinate. For **Z axis**, this value is the Z coordinate.
- **Point #1 Offset1 -** When **Y reference** or **Z reference** is selected for **Point #1 Coord**, select a reference plane or object for **2**, and then type a value for the offset distance of the point from the plane.

Point #1 Orient - Select **Offset** to define point coordinates based on the coordinate system. Select **Reference** to define point coordinates as offsets from a reference plane or object.

- **Point #1 Coord2** - When **Offset** is selected for **Point #1 Orient**, type a coordinate value for **Point #1 Coord**. For **Y axis**, this value is the Z coordinate. For **Z axis**, this value is the Y coordinate.
- **Start Pt Offset2** - When **Reference** is selected for **Point #1 Orient**, select a reference plane or object for **3**, and then type a value for the offset distance of the point from the plane.

Point #2 through Point #5 Categories

Defines subsequent points on the polyline. The number of points varies with the value of **Segment Count**. For example, **Point#1** and **Point#2** are displayed if there is one segment. **Point #1**, **Point #2**, **Point #3**, **Point #4**, and **Point #5** are displayed if there are four segments.

For **Point #2**:

Point #2 Coord - Select **Y axis**, **Z axis**, **Y reference**, or **Z reference**.

- **Point #2 Coord1** - When **Y axis** or **Z axis** is selected for **Point #2 Coord**, type a coordinate value. For **Y axis**, this value is the Y coordinate. For **Z axis**, this value is the Z coordinate.
- **Point #2 Offset1 -** When **Y reference** or **Z reference** is selected for **Point #2 Coord**, select a reference plane or object for **2**, and then type a value for the offset distance of the point from the plane.

Point #2 Orient - Select **Offset** to define point coordinates based on the coordinate system. Select **At angle** to define the line segment based on an axis and angle. Select **Parallel** to define the line segment based on a parallel line. Select **Reference** to define point coordinates as offsets from a reference plane or object.

- **Point #2 Coord2** - When **Offset** is selected for **Point #2 Orient**, type a coordinate value for **Point #2 Coord**. For **Y axis**, this value is the Z coordinate. For **Z axis**, this value is the Y coordinate.
- **Point #2 Offset2** - When **Reference** is selected for **Point #2 Orient**, select a reference plane or object for **3**, and then type a value for the offset distance of the point from the plane.
- **Point #2 Angle** - When At angle is selected for **Point #2 Orient**, select an axis (**Y axis** or **Z axis**) and type an angle to define the line starting at **Point #2**.

NOTE Options similar to those for **Point #2** are also available for **Point #3**, **Point #4**, and **Point #5**.

Other Category

Defines extension and symmetry properties.

Start Extension - Defines the distance the first line is extended before the first point.

End Extension - Defines the distance the first line is extended after the last point.

Symmetry - Defines symmetry of the polyline about the longitudinal centerline. Select **Side Y+**, **Side Y-**, or **Both Sides**.

What do you want to do?

- *Place a polyline* (on page 1114)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a polyline

Start polyline definition

1. Click **PolylineBy1to4Segments** on the **Geometric Construction Palette**.
2. In the **Auto Preview** box, select **On**.
3. Select a reference plane along the X-axis.

   A preview of the polyline appears with default values.
4. Optionally, select an offset distance from the plane in the **Ref Plane Offset** box.
5. In the **Segments Count** box, select the needed number of segments.

Define the start point

1. In the category box following **Segments Count**, select **Point #1**.
2. In the **Point #1 Coord** box, select the needed axis or reference. If a reference is selected, also select a reference plane or object for **2**.
3. If an axis is selected, type a value for **Point #1 Coord1**. If a reference is selected, type a value for **Point #1 Offset1**.
4. In the **Point #1 Orient** box, select **Offset** or **Reference**.
5. If **Offset** is selected, type a value for **Point #1 Coord2**. If **Reference** is selected, then select a reference plane or object for **3** and type a value for **Start Pt Offset**.

   The preview updates as options as changed.

Define the remaining points

1. In the category box following **Segments Count**, select **Point #2**.

2. In the **Point #2 Coord** box, select the needed axis or reference. If a reference is selected, also select a reference plane or object for **4**.
3. If an axis is selected, type a value for **Point #2 Coord1**. If a reference is selected, type a value for **Point #2 Offset1**.
4. In the **Point #2 Orient** box, select **Offset**, **Reference**, **At angle**, or **Parallel**.
5. Define values related to the selected option.
6. Repeat the steps in this section for the remaining points.

   The preview updates as options as changed.

Finish polyline creation

1. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PolylineBy1to4Segments** in the **Geometric Construction Explorer**.*
2. If needed, create additional geometric constructions.
3. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

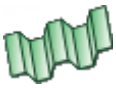
  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surfaces (Geometric Construction Palette Dialog Box)

The following geometric construction interfaces create planar and curved surfaces.

NOTE Each command supporting the 3D option displays tabs and interfaces appropriate to the command.

	**Plane from Coordinate System** - Creates a plane normal to an axis of a coordinate system at a given distance from the coordinate system origin. For more information, see *Plane from Coordinate System* (on page 1117).
	**Plane by Points** - Creates a plane from three points. For more information, see *Plane by Points* (on page 1120).

	**Plane by Positions** - Creates a plane from three points, either by selecting points or defining point coordinates. For more information, see *Plane by Positions* (on page 1123).
	**Plane by Two Points and Normal** - Creates a plane passing through two points and normal to a curve. For more information, see *Plane by Two Points and Normal* (on page 1126).
	**Surface by Curves** - Creates a multi-patch surface faired from a series of cross-section curves. For more information, see *Surface by Curves* (on page 1130).
	**Surface by Corrugation** (on page 1453) - Creates a corrugated surface by extruding a corrugation cross-section. For more information, see *Surface by Corrugation* (on page 1133).
	**Surface by Surfaces** - Creates a single surface from two or more edge-to-edge curves. For more information, see *Surface by Curves* (on page 1130).
	**Surface by Swage (on page 1465)** - Creates a swaged surface by extruding a cross-section with a swage pattern and defining the top and bottom locations. For more information, see *Surface by Swage* (on page 1145).
	**Surface by Sweeping** - Creates a surface from a set of planar cross-section curves and path curves. For more information, see *Surface by Sweeping* (on page 1153).

The following surface geometric constructions are available by selecting **More** in the **Geometric Construction** box of any geometric construction ribbon:

	Surface by Blending - Creates a surface from a set of curves while maintaining tangencies with adjacent surfaces. For more information, see *Surface by Blending* (on page 1158).
	Surface by Fillet - Creates a radius surface between two surfaces. For more information, see *Surface by Fillet* (on page 1161).
	Surface with Knuckles - Creates a multi-patch knuckled surface. For more information, see *Surface with Knuckles* (on page 1165).

Plane from Coordinate System



Tooltip: PlaneFromCS

Creates a plane normal to an axis of a coordinate system at a given distance from the coordinate system origin.

Required Input	**Output**
Coordinate system	Plane
Parameters	
Axis	
Offset from origin	
Range of the plane from the origin	

Plane from Coordinate System Ribbon

 **Select**

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

 **Add**

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

 **Delete**

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click  again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric**

Construction dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

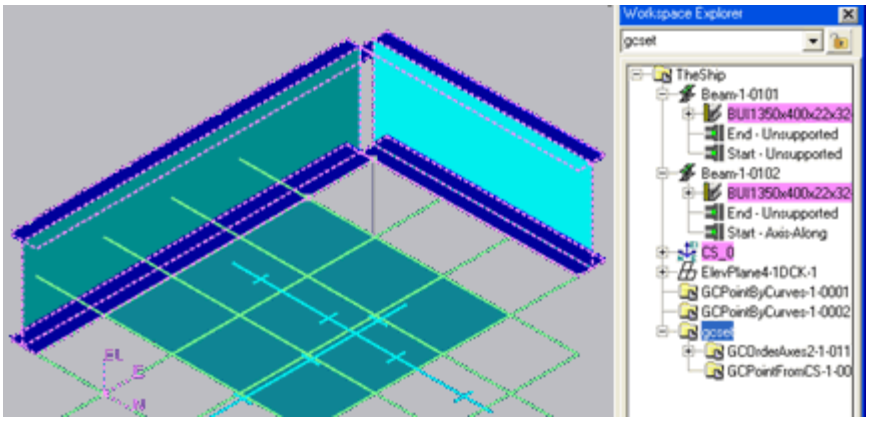
1 - Specifies the coordinate system.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Looking Axis - Specifies the normal axis of the plane. The plane is centered about the looking axis.

Offset - Defines the distance of the plane from the coordinate system origin.

Range - Defines the size of the sides of the plane. The range is centered about looking axis.

What do you want to do?

- *Place a plane by coordinate system offset* (on page 1119)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a plane by coordinate system offset

1. Click **PlaneFromCS** on the **Geometric Construction Palette**.
2. Select a coordinate system.
3. Select a value for **Looking Axis**.
4. Type a value for **Offset**.
5. Type a value for **Range**.

   A preview of the plane appears. The plane is normal to the looking axis, offset from the origin, and is centered about the looking axis.
6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PlaneFromCS** in the **Geometric Construction Explorer**.*
7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Plane by Points



Tooltip: PlaneByPoints

Creates a plane from three points. The points define three vertices of the plane, and determine both the orientation and size of the plane.

Required Input	Output
Three points	Plane

Plane by Points Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first point.

2 - Specifies the second point.

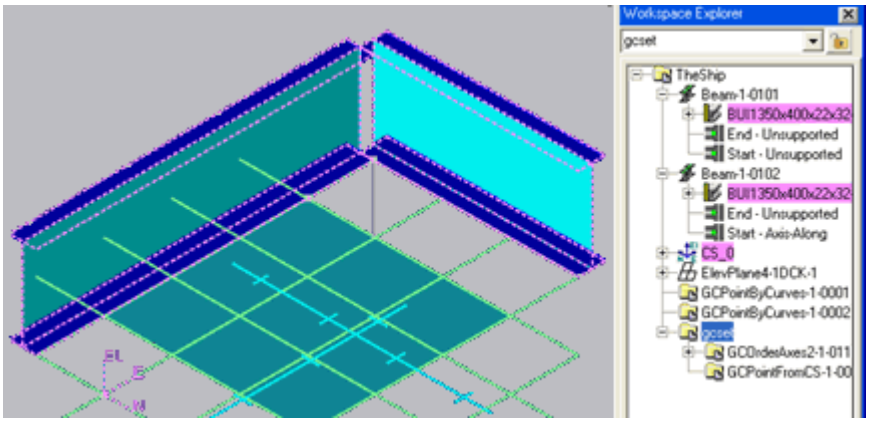
3 - Specifies the third point.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a plane by points* (on page 1122)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a plane by points

1. Create the points needed to define the plane using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **PlaneByPoints** on the **Geometric Construction Palette**.
3. Select the first point.
4. Select the second point.
5. Select the third point.

   A preview of the plane appears. Each point is a vertex of the plane.
6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PlaneByPoints** in the **Geometric Construction Explorer**.*
7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Plane by Positions



Tooltip: PlaneByPositions

Creates a plane from three points, either by selecting points or defining point coordinates. The points define an origin and two axes, are three vertices of the plane, and determine both the orientation and size of the plane.

Required Input	**Output**
Point coordinates	Plane
Optional Input	
Points	

Plane by Positions Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric**

Construction dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

(1) - Select an origin point. This option replaces the values of **x1**, **y1**, and **z1**. This value is optional.

(2) - Select a second point. This point defines a side of the plane as an axis from the origin point. This option replaces the values of **x2**, **y2**, and **z2**. This value is optional.

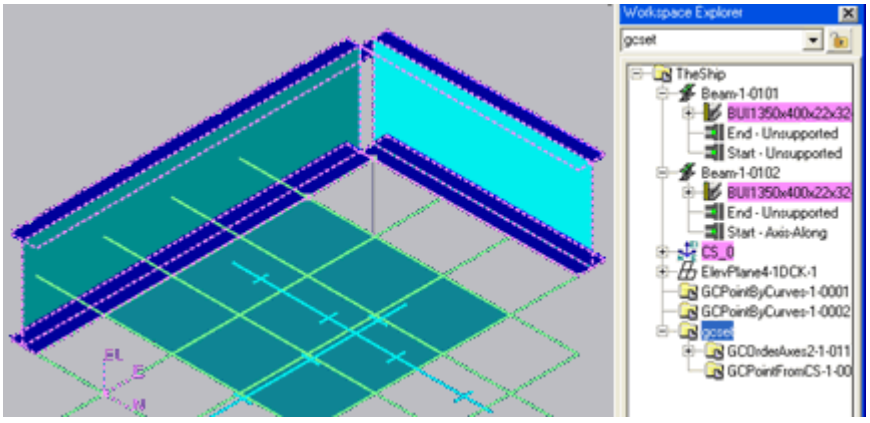
(3) - Select a third point. This point defines a second side of the plane as a second axis from the origin point. This option replaces the values of **x3**, **y3**, and **z3**. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

x1, **y1**, **z1** - Specifies the coordinates for the origin point.

x2, **y2**, **z2** - Specifies the coordinates for the second point.

x3, **y3**, **z3** - Specifies the coordinates for the third point.

What do you want to do?

- *Place a plane by coordinate positions* (on page 1125)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a plane by coordinate positions

1. If you are selecting any points instead of typing point coordinates, create the points using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **PlaneByPositions** on the **Geometric Construction Palette**.
3. Type coordinate values for the origin point in the **x1**, **y1**, and **z1** boxes. Optionally, click (1), and select a point.
4. Type coordinate values for the second point to define the first axis in the **x2**, **y2**, and **z2** boxes. Optionally, click (2), and select a point.
5. Type coordinate values for the third point to define the second axis in the **x3**, **y3**, and **z3** boxes. Optionally, click (3), and select a point.

   A preview of the plane appears. Each point is a vertex of the plane.
6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PlaneByPositions** in the **Geometric Construction Explorer**.*
7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

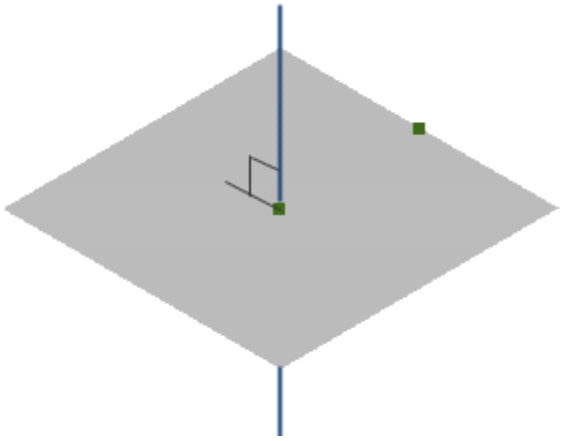
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Plane by Two Points and Normal



Tooltip: PlaneBy2PointsAndNormal

Creates a plane passing through two points and a curve. One point is on the curve and the plane is normal to the curve at that point.



Required Input	**Output**
Curve	Plane
Point on the curve (acting as an origin)	
Second point	
Parameters	
Range of the plane from the origin	

Plane by Two Points and Normal Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the curve.

2 - Specifies a point on the curve.

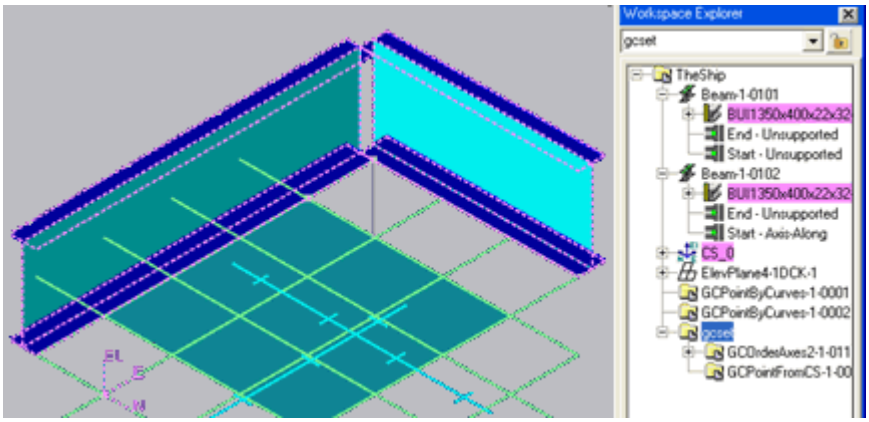
3 - Specifies the second point.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Plane Range - Defines the size of the sides of the plane. the range is centered about the point on the curve **2**. If no value is specified, the range is twice the distance between the two points.

What do you want to do?

- *Place a plane by two points normal to a curve* (on page 1129)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a plane by two points normal to a curve

1. Create the curve needed to define the plane.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)

2. Create the points needed to define the plane using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces. One point must be on the curve. The other point is not on the curve.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)

3. Click **PlaneBy2PointsAndNormal** on the **Geometric Construction Palette**.
4. Select a curve.
5. Select a point on the curve.
6. Select another point which is not on the curve.
7. Optionally, type a length for the side of the plane in the **Plane Range** box.

   A preview of the plane appears. The plane is along the two points and normal to the curve and is centered about the point on the curve.

8. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PlaneByPoints** in the **Geometric Construction Explorer**.*

9. If needed, create additional geometric constructions.
10. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surface by Curves

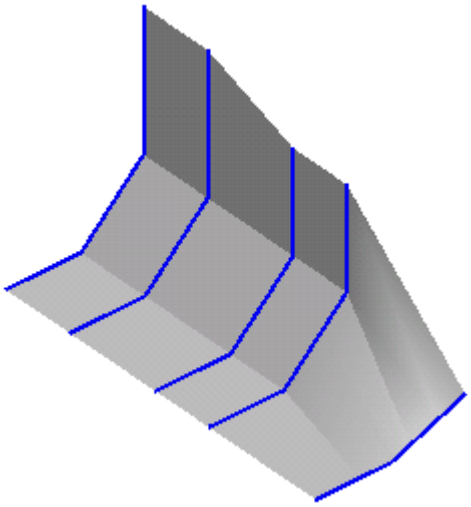


Tooltip: SurfByCurves

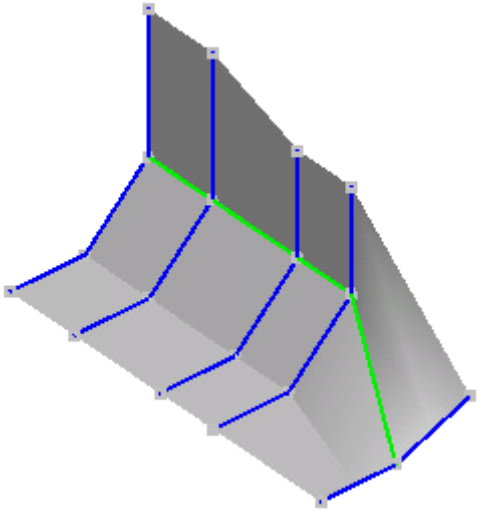
Creates a multi-patch surface *faired* (see "*fair*" on page 1456) from a series of cross-section curves, lines, or polylines.

If the cross-section curves have different quantities of segments, or you want to control the surface fairing differently than the software, you can also define a line string connecting equivalent vertices on the curves so that the surface is faired correctly from one curve to the next.

Without line string:



With line string:



Required Input	**Output**
Cross-section curves, lines, or polylines **Optional Input** Optional line string joining equivalent vertices	Surface

Surface by Curves Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the cross-section curves, lines, or polylines. Select the curves in the order they are used to create the surface.

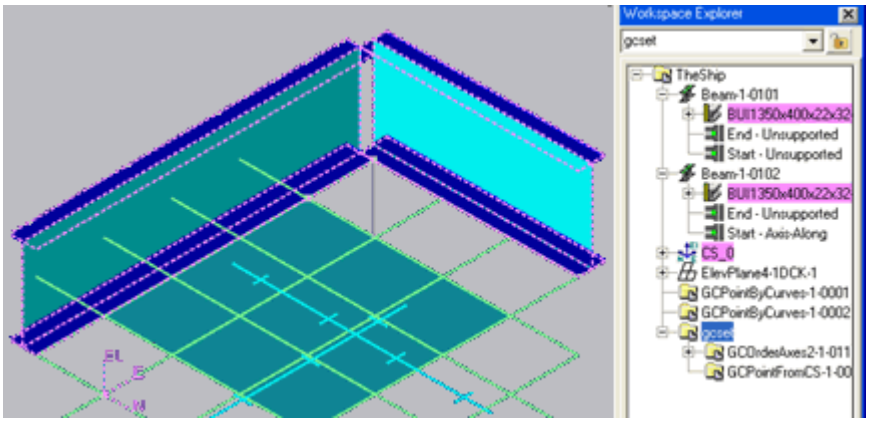
(2) - Specifies a line string connecting common vertices on the curves. This value is optional and is only needed when you want to control the fairing of the surface in a different manner than performed by the software, such as when the curves have a different number of vertices.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a surface by cross-section curves* (on page 1132)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a surface by cross-section curves

1. Create the cross-section curves, lines, or polylines needed to define the surface.

Curves (Geometric Construction Palette Dialog Box) (on page 1061)

2. If you want to use a line string to control the surface creation, create points at the curve vertices, and connect the needed points from curve to curve with a line string.

   Vertices from Curve or Surface (on page 1057)

   Line String by Points (on page 1104)

3. Click **SurfByCurves** on the **Geometric Construction Palette**.
4. Select the curves in order.
5. Optionally, click (2), and select the line string.

   A preview of the surface appears.

6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **SurfByCurves** in the **Geometric Construction Explorer**.*

7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

★ IMPORTANT

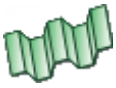
- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surface by Corrugation



Tooltip: SurfByCorrugation

Creates a *corrugated* (see "*corrugation*" on page 1453) surface by extruding a corrugation cross-section and defining the top and bottom locations.

Required Input	**Output**
Coordinate system	Corrugated surface
Reference plane	
Parameters	
Start point, end point, height, and pattern dimensions	

Surface by Corrugation Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the coordinate system.

2 - Specifies the reference plane used as a supporting surface for the corrugated surface.

3 - Specifies a starting point reference plane or object. The intersection of this plane with reference plane **2** defines the starting point of the corrugation cross-section. This option is only available when **Reference** is selected for **Start Pt Orient Type**.

4 - Specifies an ending point reference plane or object. The intersection of this plane with reference plane **2** defines the ending point of the corrugation cross-section. This option is only available when **Reference** is selected for **End Pt Orient Type**.

5 - Specifies a starting height reference plane or object. The intersection of this plane with reference plane **2** defines the bottom height of the corrugation surface. This option is only available when **Reference** is selected for **Start Height Orient**.

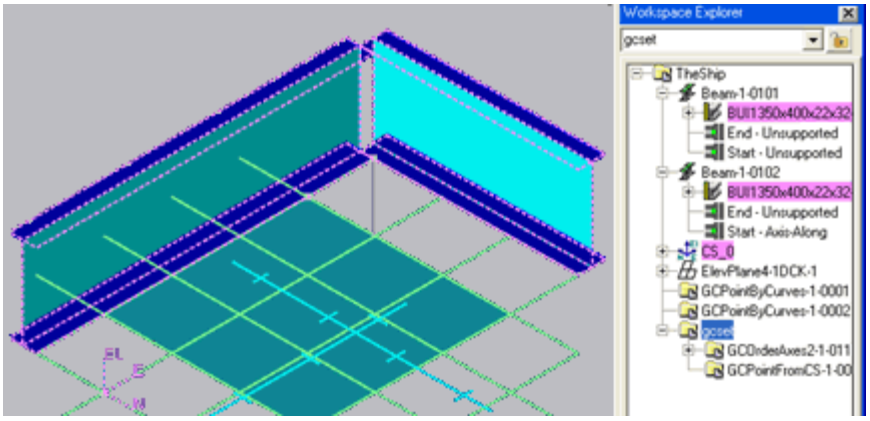
6 - Specifies a ending height reference plane or object. The intersection of this plane with reference plane **2** defines the top height of the corrugation surface. This option is only available when **Reference** is selected for **End Height Orient**.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Auto Preview - Specifies whether a preview of the corrugated surface is displayed. The preview updates as options are changed. Select **On** or **Off**.

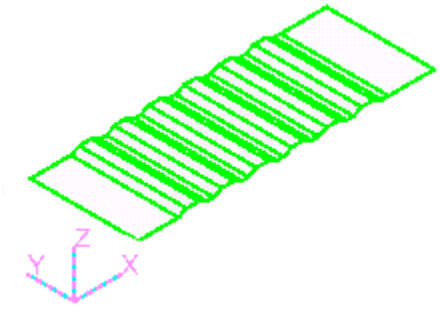
Category - Specifies the type of corrugation parameters. Select **Main**, **Start Pt**, **End Pt**, **Start Height**, **End Height**, **Pattern**, or **All**. Descriptions of the parameters for each category are below. **All** displays the properties for all categories.

Main Category

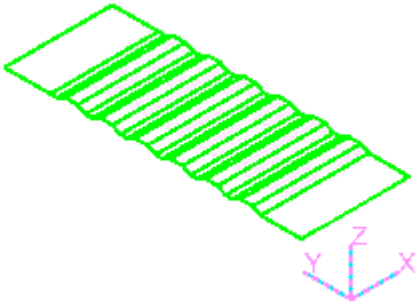
Defines general orientation properties.

Axis - Defines the orientation of the corrugated cross-section on the reference plane. Select **X axis**, **Y axis**, or **Z axis**. For example, if the reference plane is on an X-Y plane, then valid selections are **X axis** or **Y axis**.

X axis:



Y axis:



Corrugation Dir - Defines the direction of the corrugation cross-section with respect to the reference plane. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. For example, if the reference plane is on an X-Y plane, then valid selections are **+z** or **-z**.

+z:



-z:



Ref Plane Offset - Defines the offset of the corrugated surface from the reference plane.

Start Pt Category

Defines the starting point of the corrugation cross-section.

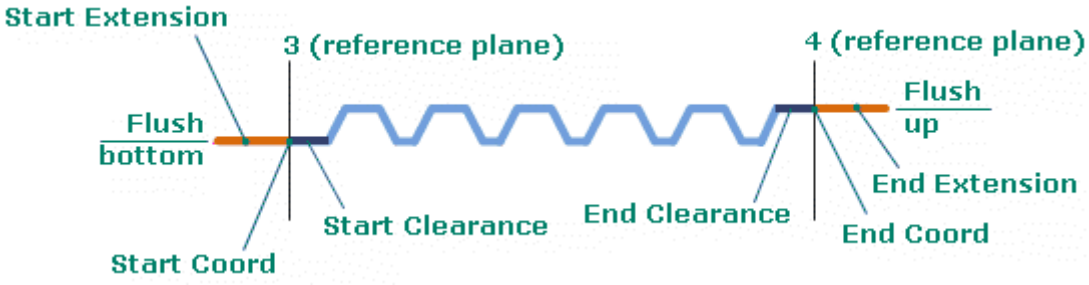
Start Pt Orient Type - Select **Offset** to define point coordinates based on the coordinate system. Select **Reference** to define point coordinates as the intersection of a reference plane or object (by clicking **3**) with reference plane **2**.

Start Coord - Defines a coordinate value for the starting point. Type a value. This option is only available when **Offset** is selected for **Start Pt Orient Type**.

Start Clearance - Defines the distance between the starting point and the first bend of the corrugation. Type a value.

Start Extension - Defines a value for the distance the surface is extended before the starting point. Type a value.

Start Pos Type - Defines orientation of the surface before the first bend of the corrugation. Select **Flush up** or **Flush bottom**.



End Pt Category

Defines the ending point of the corrugation cross-section.

End Pt Orient Type - Select **Offset** to define point coordinates based on the coordinate system. Select **Reference** to define point coordinates as the intersection of a reference plane or object (by clicking **4**) with reference plane **2**.

End Coord - Defines a coordinate value for the ending point. Type a value. This option is only available when **Offset** is selected for **End Pt Orient Type**.

End Clearance - Defines the distance between the ending point and the last bend of the corrugation. Type a value.

End Extension - Defines a value for the distance the surface is extended after the ending point. Type a value.

End Pos Type - Defines orientation of the surface after the last bend of the corrugation. Select **Flush up** or **Flush bottom**.

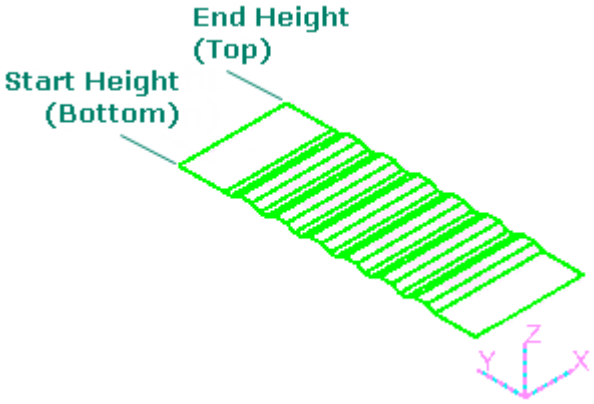
Start Height Category

Defines the bottom location of the corrugation surface.

Start Height Orient - Select **Offset** to define height coordinates based on the coordinate system. Select **Reference** to define height coordinates as the intersection of a reference plane or object with reference plane **2**.

Start Height - Defines a coordinate value for the bottom of the corrugated surface. Type a value. This option is only available when **Offset** is selected for **Start Height Orient**.

Start Height Offset - Defines the offset of the bottom of the corrugated surface from reference plane **5**. This option is only available when **Reference** is selected for **Start Height Orient**.



End Height Category

Defines the top location of the corrugation surface.

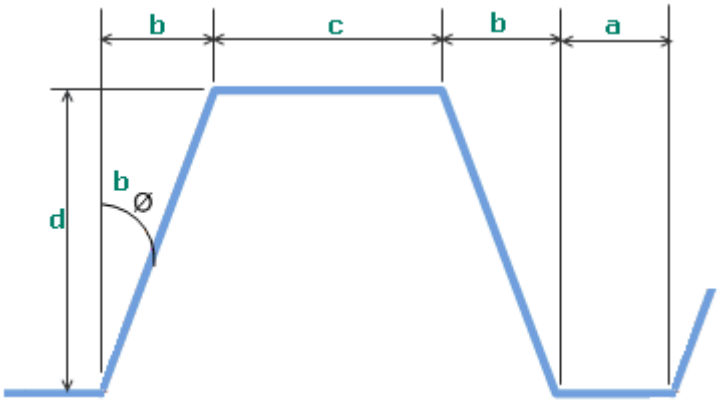
End Height Orient - Select **Offset** to define height coordinates based on the coordinate system. Select **Reference** to define height coordinates as the intersection of a reference plane or object with reference plane **2**.

End Height - Defines a coordinate value for the top of the corrugated surface. Type a value. This option is only available when **Offset** is selected for **End Height Orient**.

End Height Offset - Defines the offset of the top of the corrugated surface from reference plane **6**. This option is only available when **Reference** is selected for **End Height Orient**.

Pattern Category

Defines the corrugation pattern.



a - The distance between corrugations at the bottom of the cross-section.

b - The slope width. This option is only available when **Offset** is selected for **b Type**.

b as angle - The slope angle. This option is only available when **Angle** is selected for **b Type**.

c - Width of the corrugation at the top of the cross-section.

d - Corrugation height.

What do you want to do?

- *Place a corrugated surface* (on page 1139)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

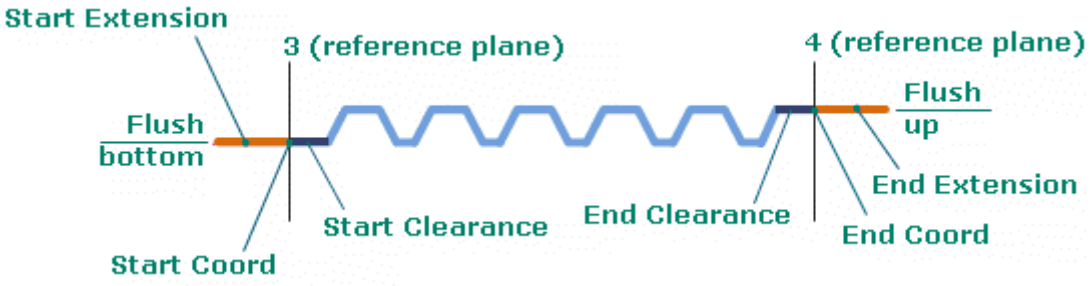
Place a corrugated surface

Start corrugation definition

1. Click **SurfByCorrugation** on the **Geometric Construction Palette**.
2. In the **Auto Preview** box, select **On**.
3. Select a coordinate system.
4. Select a reference plane.

   A preview of the corrugated surface appears, oriented on the reference plane and using default values.

Define the starting point of the corrugation cross-section



1. In the **Category** box, select **Start Pt**.
2. If you select **Offset** in the **Start Pt Orient Type** box:
   - Type values for **Start Coord**, **Start Clearance**, and **Start Extension**.
   - Select a value for **Start Pos Type**.

   The start of the corrugation cross-section updates in the preview as options as changed.
3. If you select **Reference** in the **Start Pt Orient Type** box:
   - Click **3**, and select a reference plane or object.
   - Type values for **Start Clearance** and **Start Extension**.
   - Select a value for **Start Pos Type**.

   The start of the corrugation cross-section updates in the preview as options as changed.

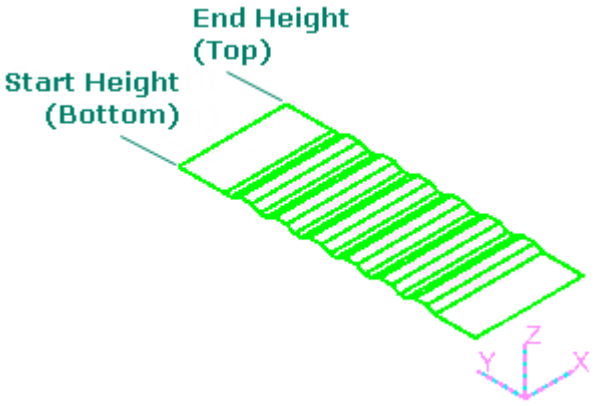
Define the ending point of the corrugation cross-section

1. In the **Category** box, select **End Pt**.
2. If you select **Offset** in the **End Pt Orient Type** box:
   - Type values for **End Coord**, **End Clearance**, and **End Extension**.
   - Select a value for **End Pos Type**.

   The end of the corrugation cross-section updates in the preview as options as changed.
3. If you select **Reference** in the **Start Pt Orient Type** box:
   - Click **4**, and select a reference plane or object.
   - Type values for **End Clearance** and **End Extension**.
   - Select a value for **End Pos Type**.

   The end of the corrugation cross-section updates in the preview as options as changed.

Define the bottom and top of the corrugated surface

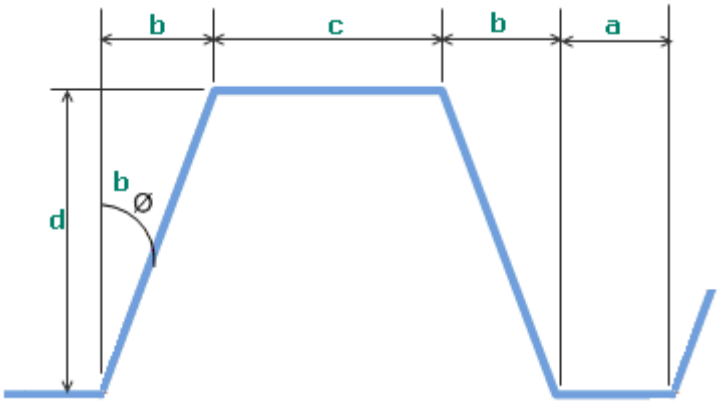


1. In the **Category** box, select **Start Height**.
2. If you select **Offset** in the **Start Height Orient** box, type a value for **Start Height**.
3. If you select **Reference** in the **Start Height Orient** box, click **5**, and select a reference plane or object, and type a value for **Start Height Offset**.

   The bottom of the corrugated surface updates in the preview as options as changed.
4. In the **Category** box, select **End Height**.
5. If you select **Offset** in the **End Height Orient** box, type a value for **End Height**.
6. If you select **Reference** in the **End Height Orient** box, click **6**, and select a reference plane or object, and type a value for **End Height Offset**.

   The top of the corrugated surface updates in the preview as options as changed.

Define the corrugation pattern



1. In the **Category** box, select **Pattern**.
2. Type values for **a**, **c**, and **d**.
3. If you select **Offset** in the **b Type** box, type a value for **b**.
4. If you select **Angle** in the **b Type** box, type a value for **b as angle**.

   The corrugation pattern updates in the preview as options as changed.

Finish surface creation

1. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **SurfByCorrugation** in the **Geometric Construction Explorer**.*
2. If needed, create additional geometric constructions.
3. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surface by Surfaces



Tooltip: SurfBySurfaces

Creates a single surface from two or more edge-to-edge curves.

Required Input	Output
Two or more surfaces	Surface

Surface by Surfaces Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

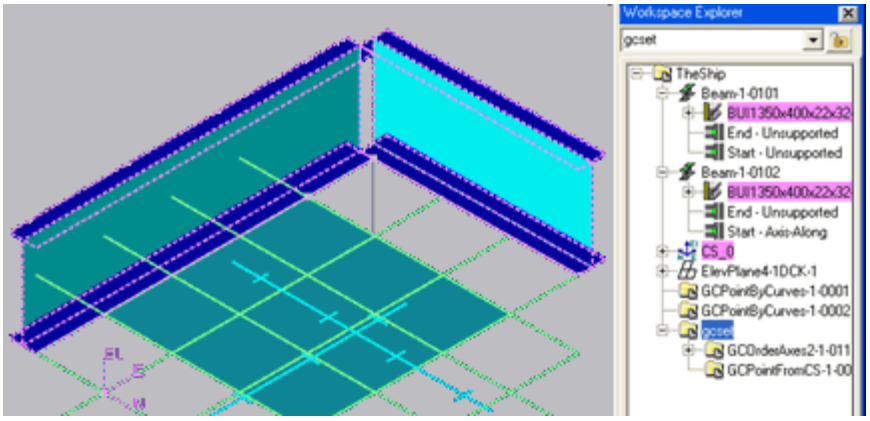
1 - Specifies two or more surfaces to merge. Each surface must share a common edge with at least one of the other surfaces.

Color Coding

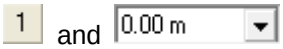
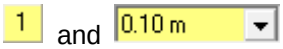
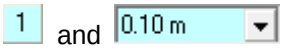
For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background color when there is no input.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

-  - No background color when you keep the value of the original geometric construction.
-  - Yellow background when you change a value.
-  - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Merge surfaces into a single surface* (on page 1144)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Merge surfaces into a single surface

1. Click **SurfbySurfaces** on the **Geometric Construction Palette**.
2. Select the surfaces to combine. Each surface must share a common edge with another of the surfaces.
3. Click **Continue**.

   *Temporary geometry of the merged surfaces is displayed in the graphic view and as **SurfbySurfaces** in the **Geometric Construction Explorer**.*
4. If needed, create additional geometric constructions.
5. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surface by Swage



Tooltip: SurfBySwage

Creates a *swaged* (see "*swage*" on page 1465) surface by extruding a cross-section with a swage pattern and defining the top and bottom locations. A swage cross-section can be very similar to the regular, elongated-V pattern of a corrugated surface, but more frequently has radius or V bends and an irregular pattern.

Required Input	**Output**
Coordinate system	Swaged surface
Reference plane	
Parameters	
Start point, end point, height, and pattern dimensions	

Surface by Swage Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the coordinate system.

2 - Specifies the reference plane used as a supporting surface for the swaged surface.

3 - Specifies a starting point reference plane or object. The intersection of this plane with reference plane **2** defines the starting point of the swage cross-section. This option is only available when **Reference** is selected for **Start Pt Orient Type**.

4 - Specifies an ending point reference plane or object. The intersection of this plane with reference plane **2** defines the ending point of the swage cross-section. This option is only available when **Reference** is selected for **End Pt Orient Type**.

5 - Specifies a starting height reference plane or object. The intersection of this plane with reference plane **2** defines the bottom height of the swage surface. This option is only available when **Reference** is selected for **Start Height Orient**.

6 - Specifies a ending height reference plane or object. The intersection of this plane with reference plane **2** defines the top height of the swage surface. This option is only available when **Reference** is selected for **End Height Orient**.

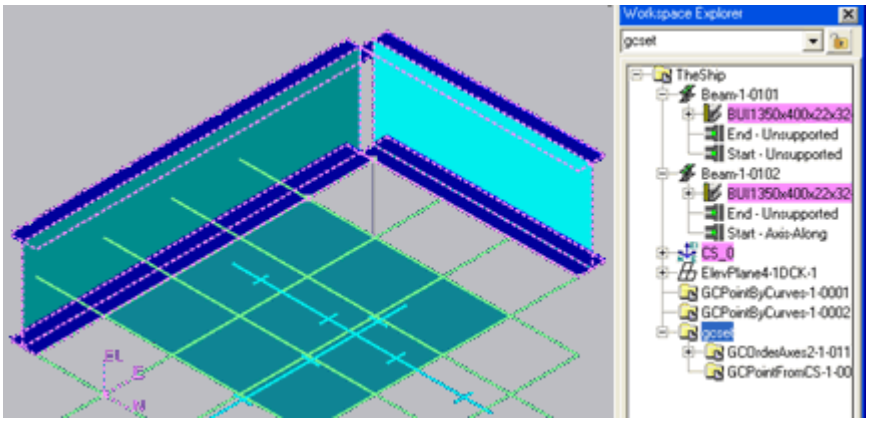
7 - Specifies the center locations for swage features when **Non-Regular** is selected for **Swage Pattern**.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background color when there is no input.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Auto Preview - Specifies whether a preview of the swaged surface is displayed. The preview updates as options are changed. Select **On** or **Off**.

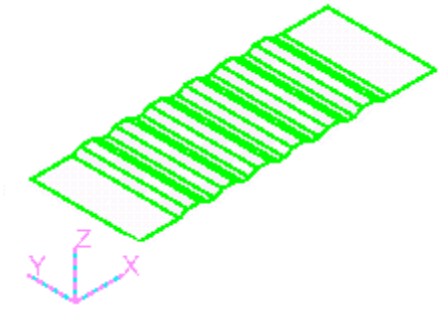
Category - Specifies the type of swage parameters. Select **Main**, **Start Pt**, **End Pt**, **Start Height**, **End Height**, **Pattern**, or **All**. Descriptions of the parameters for each category are below. **All** displays the properties for all categories.

Main Category

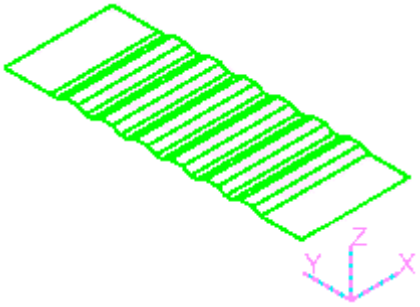
Defines general orientation properties.

Along Axis - Defines the orientation of the swage cross-section on the reference plane. Select **X axis**, **Y axis**, or **Z axis**. For example, if the reference plane is on an X-Y plane, then valid selections are **X axis** or **Y axis**.

X axis:

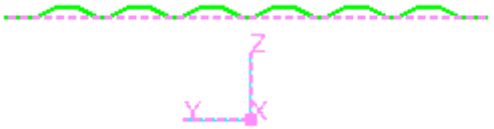


Y axis:



Swage Dir - Defines the direction of the swage cross-section with respect to the reference plane. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. For example, if the reference plane is on an X-Y plane, then valid selections are **+z** or **-z**.

+z:



-z:



Ref Plane Offset - Defines the offset of the swage surface from the reference plane.

Start Pt Category

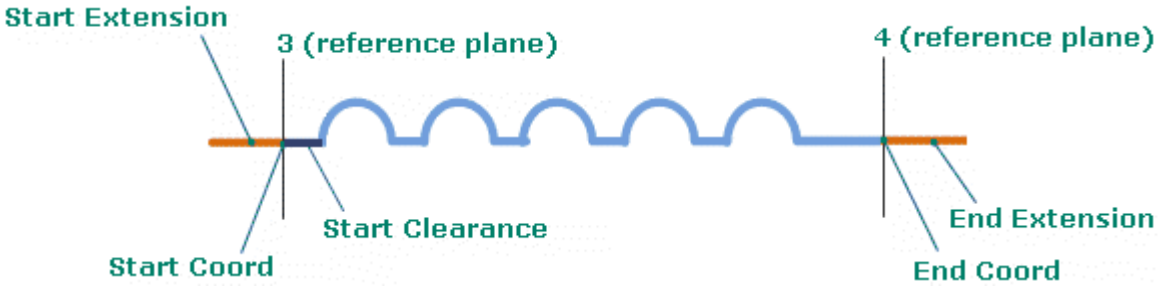
Defines the starting point of the swage cross-section.

Start Pt Orient Type - Select **Offset** to define point coordinates based on the coordinate system. Select **Reference** to define point coordinates as the intersection of a reference plane or object (by clicking **3**) with reference plane **2**.

Start Coord - Defines a coordinate value for the starting point. Type a value. This option is only available when **Offset** is selected for **Start Pt Orient Type**.

Start Clearance - Defines the distance between the starting point and the first bend of the swage. Type a value.

Start Extension - Defines a value for the distance the surface is extended before the starting point. Type a value.



End Pt Category

Defines the ending point of the swage cross-section.

End Pt Orient Type - Select **Offset** to define point coordinates based on the coordinate system. Select **Reference** to define point coordinates as the intersection of a reference plane or object (by clicking **4**) with reference plane **2**.

End Coord - Defines a coordinate value for the ending point. Type a value. This option is only available when **Offset** is selected for **End Pt Orient Type**.

End Extension - Defines a value for the distance the surface is extended after the ending point. Type a value.

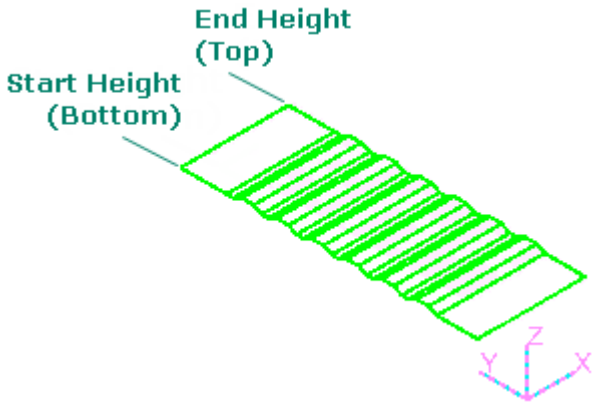
Start Height Category

Defines the bottom location of the swaged surface.

Start Height Orient - Select **Offset** to define height coordinates based on the coordinate system. Select **Reference** to define height coordinates as the intersection of a reference plane or object with reference plane **2**.

Start Height - Defines a coordinate value for the bottom of the swaged surface. Type a value. This option is only available when **Offset** is selected for **Start Height Orient**.

Start Height Offset - Defines the offset of the bottom of the swaged surface from reference plane **5**. This option is only available when **Reference** is selected for **Start Height Orient**.



End Height Category

Defines the top location of the swaged surface.

End Height Orient - Select **Offset** to define height coordinates based on the coordinate system. Select **Reference** to define height coordinates as the intersection of a reference plane or object with reference plane **2**.

End Height - Defines a coordinate value for the top of the swaged surface. Type a value. This option is only available when **Offset** is selected for **End Height Orient**.

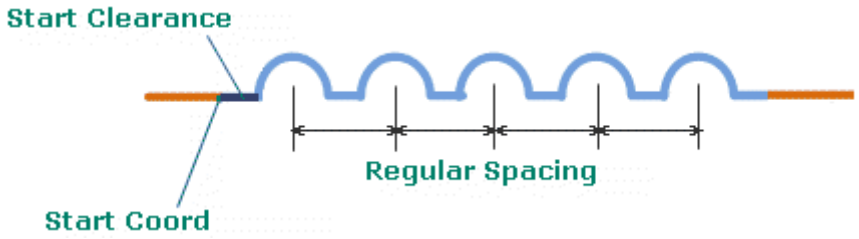
End Height Offset - Defines the offset of the top of the swaged surface from reference plane **6**. This option is only available when **Reference** is selected for **End Height Orient**.

Pattern Category

Defines the swage feature shape and spacing.

Swage Pattern - Specifies the pattern for swage spacing:

- **Regular** - Places features at the center-to-center interval specified by **Spacing**. The edge of the first feature is at the **Start Clearance** location.

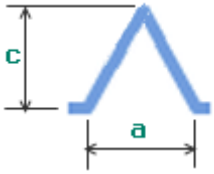


- **Non-Regular** - Places the center of features at the reference planes selected for **7**.

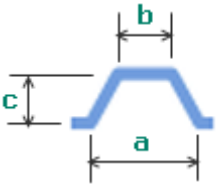


Feature Type - Specifies the shape of the swage features. Select **V**, **Elongated V**, or **Arc**.

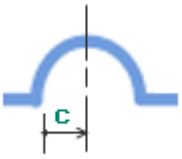
V:



Elongated V:



Arc:



a - The width at the bottom of a **V** or **Elongated V** feature.

b - The width at the top of an **Elongated V** feature.

c - The height for a **V** or **Elongated V** feature. The radius for an **Arc** feature.

Spacing - The center-to-center distance between features. This option is only available when **Regular** is selected for **Swage Pattern**.

What do you want to do?

- *Place a swaged surface* (on page 1151)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

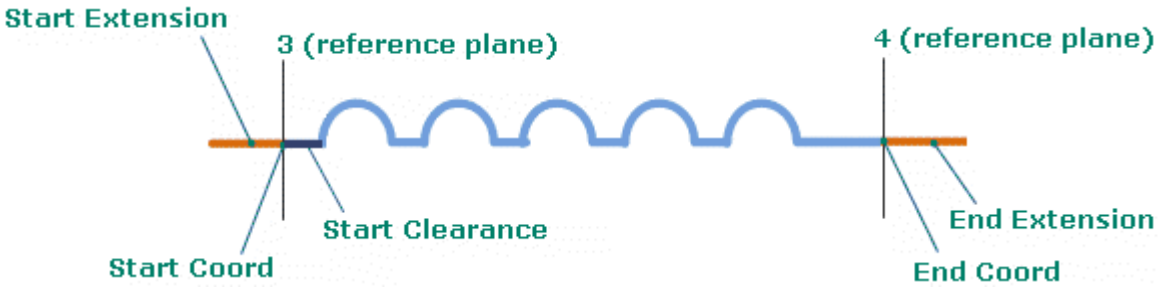
Place a swaged surface

Start swage definition

1. Click **SurfBySwage** on the **Geometric Construction Palette**.
2. In the **Auto Preview** box, select **On**.
3. Select a coordinate system.
4. Select a reference plane.

   A preview of the swaged surface appears, oriented on the reference plane.

Define the starting point of the swage cross-section



1. In the **Category** box, select **Start Pt**.
2. If you select **Offset** in the **Start Pt Orient Type** box, type values for **Start Coord**, **Start Clearance**, and **Start Extension**.

   The start of the swage cross-section updates in the preview as options as changed.
3. If you select **Reference** in the **Start Pt Orient Type** box:
   - Click **3**, and select a reference plane or object.
   - Type values for **Start Clearance** and **Start Extension**.

   The start of the swage cross-section updates in the preview as options as changed.

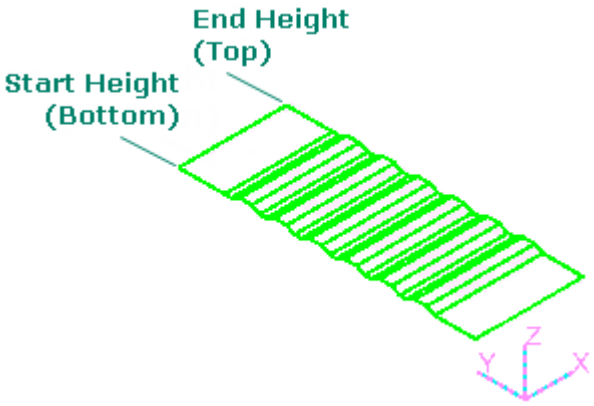
Define the ending point of the swage cross-section

1. In the **Category** box, select **End Pt**.
2. If you select **Offset** in the **End Pt Orient Type** box, type values for **End Coord** and **End Extension**.

   The end of the swage cross-section updates in the preview as options as changed.
3. If you select **Reference** in the **Start Pt Orient Type** box:
   - Click **4**, and select a reference plane or object.
   - Type a value for **End Extension**.

   The end of the swage cross-section updates in the preview as options as changed.

Define the bottom and top of the swaged surface



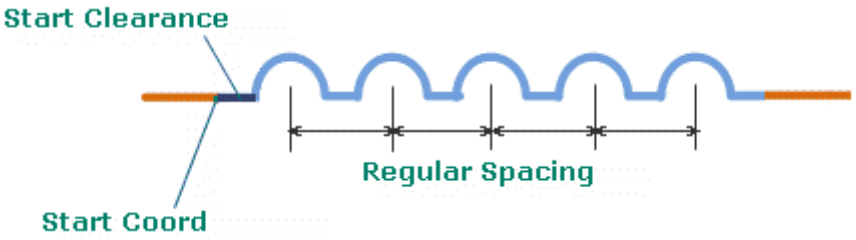
1. In the **Category** box, select **Start Height**.
2. If you select **Offset** in the **Start Height Orient** box, type a value for **Start Height**.
3. If you select **Reference** in the **Start Height Orient** box, click **5**, and select a reference plane or object, and type a value for **Start Height Offset**.

   The bottom of the swaged surface updates in the preview as options as changed.
4. In the **Category** box, select **End Height**.
5. If you select **Offset** in the **End Height Orient** box, type a value for **End Height**.
6. If you select **Reference** in the **End Height Orient** box, click **6**, and select a reference plane or object, and type a value for **End Height Offset**.

   The top of the swaged surface updates in the preview as options as changed.

Define the swage feature and spacing pattern

1. In the **Category** box, select **Pattern**.
2. In the **Swage Pattern** box, select the type of spacing.
   - If you select **Regular**, type a value for **Spacing**.

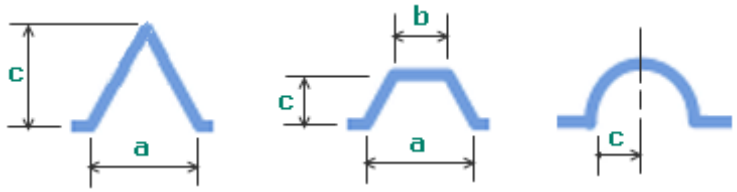


   - If you select **Non-Regular**, click **7**, and select one or more reference planes.



3. In the **Feature Type** box, select the type of swage feature.
4. Type values for **a**, **b**, and **c**, as appropriate for the selected feature type.

V: **Elongated V**: **Arc**:



The swage pattern updates in the preview as options as changed.

Finish surface creation

1. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **SurfBySwage** in the **Geometric Construction Explorer**.*

2. If needed, create additional geometric constructions.
3. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surface by Sweeping



Tooltip: SurfBySweeping

Creates a surface from a set of planar curves (such as curves on plate edges) and optional path curves.

The cross-section curves:

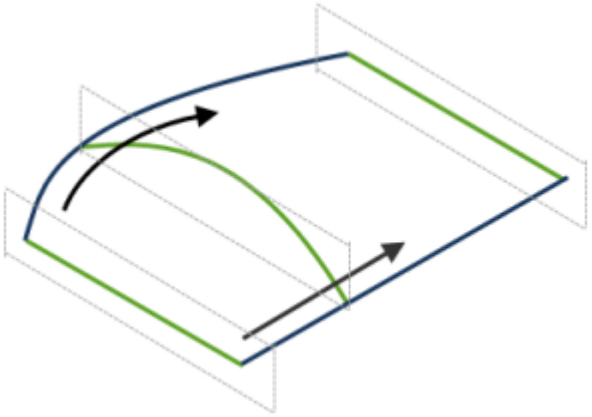
- Must be planar, but do not need to be coplanar
- Must be oriented in the same direction to avoid a twist in the surface
- Must each have the same number of vertices

- Can be open or closed
- Can touch at some vertices

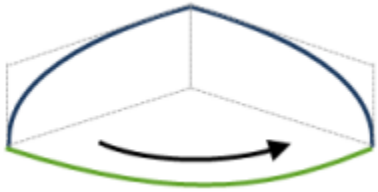
The paths curves:

- Do not need to be planar
- Must touch the cross-section curves unless there is only one path curve
- Must each have the same number of vertices

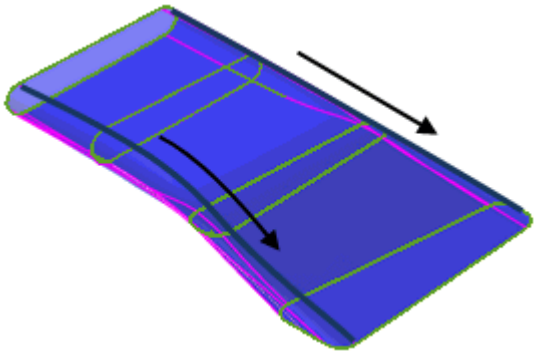
The following example shows planar cross-sections curves swept along two path curves:



The following example shows planar cross-sections curves touching at end vertices, and swept along a single path curve:



The following example shows closed cross-sections swept along two path curves. A capping surface is created on the left end:



Required Input	Output
Planar curves	Surface
Optional Input	Capping surface at one or both ends
Path curves	
Parameter	
Capping faces	

Surface by Sweeping Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the cross-section curves.

(2) - Specifies one or more path curves. These curves control the creation of the surface when multiple paths through the cross-section curves are possible. This value is optional.

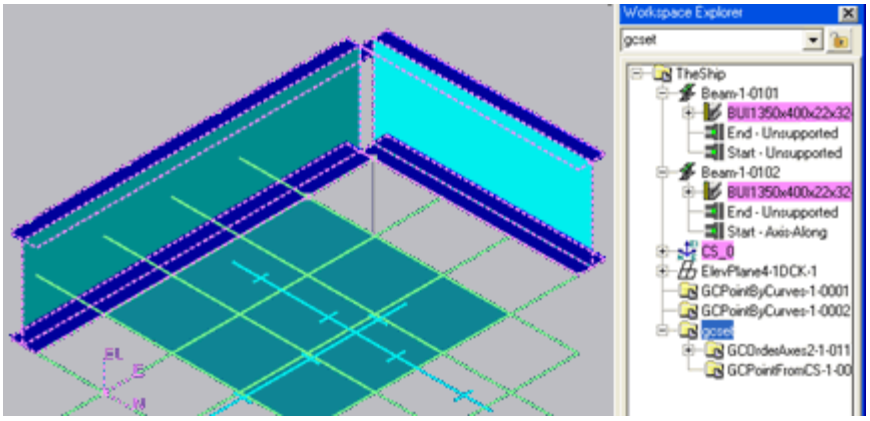
Capping Faces - Specifies surfaces capping the ends of the swept surface. This option is valid with closed cross-section curves. Select **No capping face**, **Capping face at start**, **Capping face at end**, or **Both capping faces**. Capping surfaces are merged with the main surface.

Color Coding

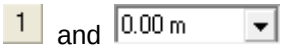
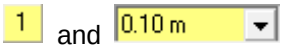
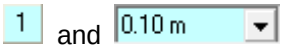
For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background color when there is no input.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

-  - No background color when you keep the value of the original geometric construction.
-  - Yellow background when you change a value.
-  - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a surface by sweeping between planar curves* (on page 1157)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a surface by sweeping between planar curves

1. Create the planar cross-section curves needed to define the surface. Optionally, create one or more path curves to control the sweep of the surface through the curves.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)
2. Click **SurfBySweeping** on the **Geometric Construction Palette**.
3. Select the cross-section curves in order.
4. Optionally, click (2), and select the path curves.

   A preview of the surface appears.
5. If the cross-section surfaces are closed, select the capping surfaces to add.
6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **SurfBySweeping** in the **Geometric Construction Explorer**.*
7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

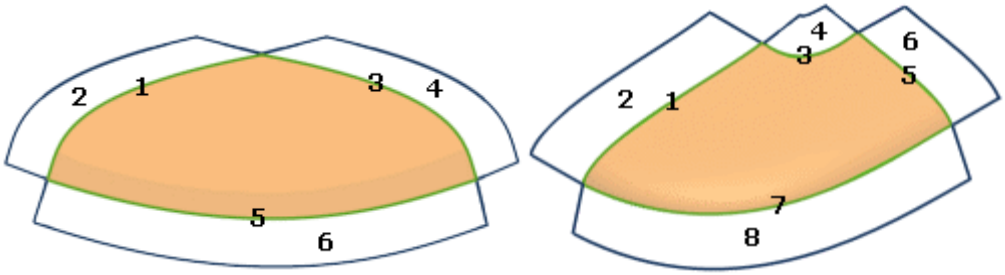
  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surface by Blending



Tooltip: SurfByBlending

Creates a surface from a set of curves while maintaining tangencies with adjacent surfaces. Two to five curve and adjacent surface combinations are required. The curves and adjacent surfaces must be selected in sequential order and create a closed contour, as shown in the following examples.



Required Input	Output
Curves and surfaces adjacent to the curves	Surface

Surface by Blending Ribbon

 **Select**

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

 **Add**

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

 Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first curve, typically at the edge of a surface.

2 - Specifies the surface adjacent to the first curve.

3 - Specifies the second curve.

4 - Specifies the surface adjacent to the first curve.

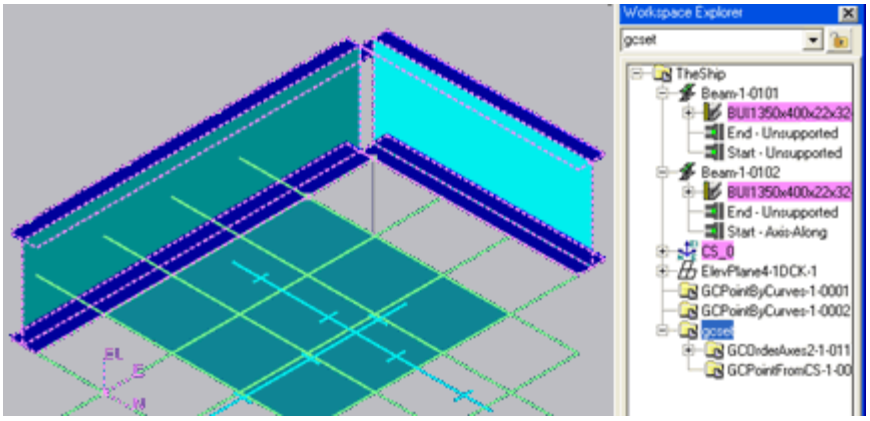
5 through **10** - Specifies additional curve and adjacent surface combinations used to create a closed contour. These values are optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

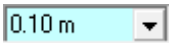
- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.

-  and  - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a surface blended between adjacent curves and surfaces* (on page 1160)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a surface blended between adjacent curves and surfaces

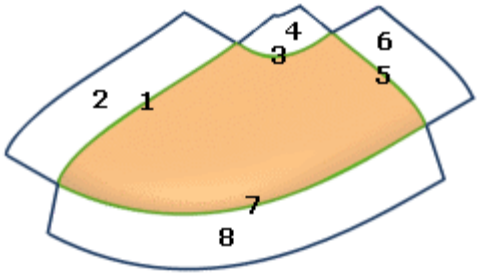
1. Create the curves needed to define the surface. These curves are typically at the edges of the adjacent surfaces.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)
2. On the geometric construction ribbon, click **More** in the **Geometric Construction** box.

   *The **Select Geometric Construction** dialog box appears.*
3. Select **SurfByBlending** and click **OK**.
4. Select the first curve.
5. Select the surface adjacent to the curve.
6. Select the second curve.
7. Select the surface adjacent to the second curve.
8. Continue select curves and surfaces, up to five pairs, until a closed contour of curves is formed.

   A preview of the surface appears.
9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **SurfByBlending** in the **Geometric Construction Explorer**.*



10. If needed, create additional geometric constructions.
11. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

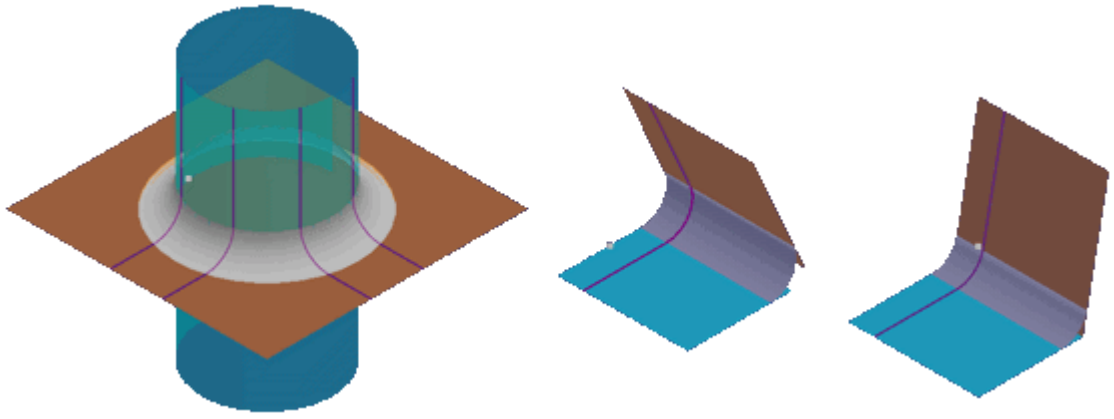
  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surface by Fillet

Tooltip: SurfByFillet

Creates a radius surface between two surfaces. The supporting surfaces must not be coplanar and do not need to touch. The radius of the fillet surface must be large enough to fill the gap between surfaces that do not touch, but smaller than the two surfaces. In the examples below, the fillet surface is in gray.



Required Input	Output
Two non-coplanar surfaces	Radius surface
Parameter	
Radius	

Surface by Fillet Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first surface.

2 - Specifies the second surface.

3 - Specifies a tracking point used to control the placement of the fillet surface.

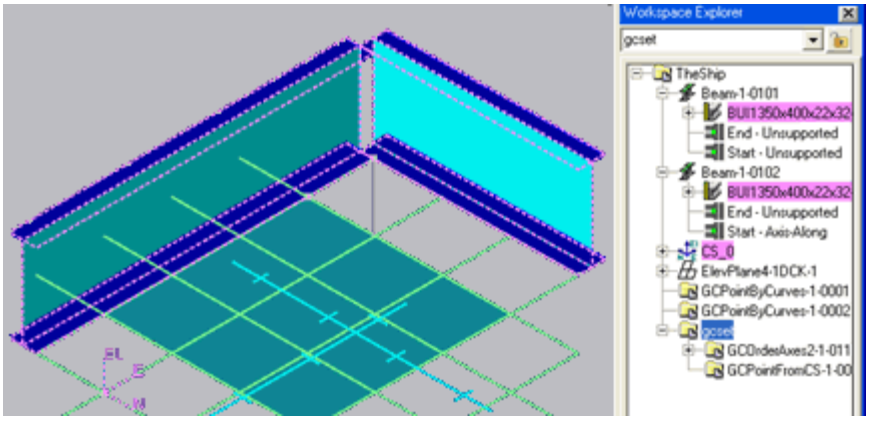
NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

Color Coding

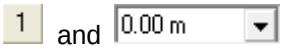
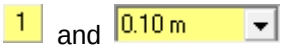
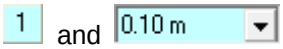
For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background color when there is no input.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

-  - No background color when you keep the value of the original geometric construction.
-  - Yellow background when you change a value.
-  - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Radius - Specifies the radius of the fillet surface.

NOTES

- If the surfaces do not touch, the radius must be large enough to create the fillet surface.
- If the radius must be smaller than supporting surfaces to create the fillet surface.

What do you want to do?

- *Place a fillet surface between two surfaces* (on page 1164)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a fillet surface between two surfaces

1. Create the point needed as the tracking point to locate the fillet surface by using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

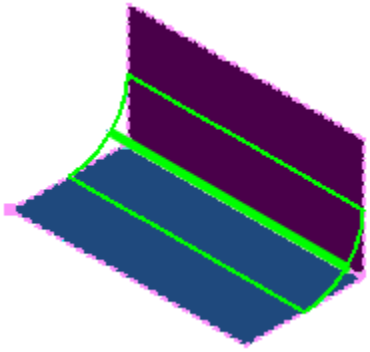
   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. On the geometric construction ribbon, click **More** in the **Geometric Construction** box.

   *The **Select Geometric Construction** dialog box appears.*
3. Select **SurfByFillet** and click **OK**.
4. Select the first surface.
5. Select the second surface.
6. Select the tracking point.

   A preview of the fillet surface appears.



7. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **SurfByFillet** in the **Geometric Construction Explorer**.*
8. If needed, create additional geometric constructions.

9. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Surface with Knuckles



Tooltip: SurfWithKnuckles

Creates a single multi-patch knuckled surface.

Required Input	**Output**
Surface	Surface
Optional Input	
Knuckles curves to remove	
Knuckles curves to add	
Surface angle at which a knuckle curve is created	

Surface with Knuckles Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies a surface.

(2) - Specifies curves to add to the surface as knuckle curves. This value is optional.

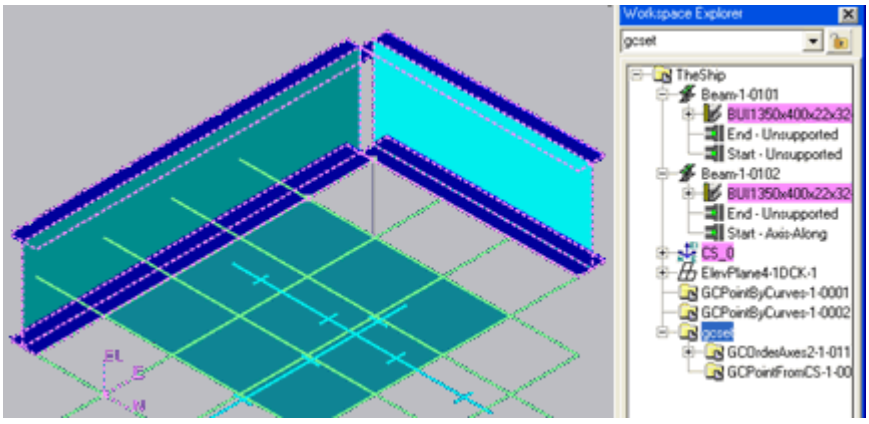
(3) - Specifies curves to remove from the surface as knuckle curves. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Max dev angle - Specifies the angle in a surface bend at which the software creates a knuckle curve.

What do you want to do?

- *Place a knuckled surface* (on page 1167)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a knuckled surface

1. Create the surface needed to define the knuckled surface.

   Surfaces (Geometric Construction Palette Dialog Box) (on page 1115)
2. Optionally, create curves to use as knuckle curves.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)
3. On the geometric construction ribbon, click **More** in the **Geometric Construction** box.

   *The **Select Geometric Construction** dialog box appears.*
4. Select **SurfWithKnuckles**, and click **OK**.

   *The **SurfWithKnuckles** ribbon appears.*
5. Select the surface.
6. Optionally, click (2), and select curves to add to the surface as knuckle curves.
7. Optionally, click (3), and select curves to remove from the surface as knuckle curves.

8. Type the maximum angle for a surface bend at which the software creates a knuckle curve.
9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **SurfWithKnuckles** in the **Geometric Construction Explorer**.*
10. If needed, create additional geometric constructions.
11. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Coordinate Systems (Geometric Construction Palette Dialog Box)

The following geometric construction interfaces create coordinate systems.

NOTE Each command supporting the 3D option displays tabs and interfaces appropriate to the command.

	**Coordinate System by Lines** - Creates a coordinate system from two lines. The first line defines the first axis. The second line defines the general direction of the second axis. For more information, see *Coordinate System by Lines* (on page 1169).
	**Coordinate System by Points** - Creates a coordinate system from three points. The first point defines the origin. The second point defines the direction of the first axis. The third point defines the general direction of the second axis. For more information, see *Coordinate System by Points* (on page 1172).

Coordinate System by Lines



Tooltip: CSByLines

Creates a coordinate system from two lines. The first line defines the first axis. The second line defines the general direction of the second axis. The origin is at the intersection of the lines.

NOTE This geometric construction only works with lines, not line strings or curves.

Required Input	**Output**
Lines for first and second axes	Coordinate system
Parameters	
Initial axes to create, such as (x,y) or (y,z)	
Direction of third axis	

Coordinate System by Lines Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies a line for the first axis. You must select a line. You cannot select a line string or a curve.

2 - Specifies a line for the second axis. You must select a line. You cannot select a line string or a curve.

(3) - Specifies a tracking point used to control the direction of axis placement when an ambiguity exists with one of the lines selected for **1** and **2**. The tracking point is used with the value selected by **Track Flag**. This value is optional.

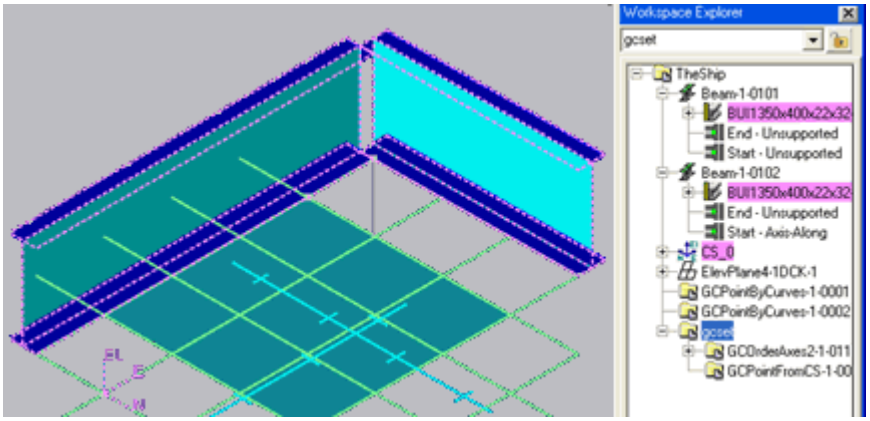
NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Assign Axes To - Specifies the axes to define by the selected lines. The first value defines the axis for line **1**. The second value defines the axis for line **2**. Select **(x,y)**, **(y,z)**, **(z,x)**, **(y,x)**, **(z,y)**, or **(x,z)**.

Coordinate System - Specifies the direction of the third axis. Select **Direct** to use the right-hand rule to define the third axis. Select **Indirect** to define the third axis in the opposite direction.

Track Flag - Controls how the tracking point (3) is used when there is an axis ambiguity with one of the lines selected for **1** and **2**. Select **Near** to use the axis direction towards the tracking point. Select **Far** to use the axis direction away from the tracking point.

What do you want to do?

- *Place a coordinate system by lines* (on page 1171)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a coordinate system by lines

1. Create the lines needed to define the coordinate system. Two lines are needed for the direction of the first axis and the general direction of the second axis on the (first axis, second axis) plane. Create the lines by using another geometric construction line interface.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)
2. Click **CSByLines** on the **Geometric Construction Palette**.
3. In the **Assign Axes To** box, select the initial axes and plane to define.
4. In the **Coordinate System** box, select **Direct**.
5. Select the first line for the direction of the first axis.
6. Select the second line for the general direction of the second axis.

   A preview of the coordinate system appears. The direction of the third axis is determined by the right-hand rule.
7. If there are multiple solutions for the first or second axis, such as when a line extends beyond the origin, click (3), select a tracking point, and select the needed value for **Track Flag**.

8. If the third axis is not in the needed direction, select **Indirect** in the **Coordinate System** box.

   The direction of the third axis is flipped.

9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CSByPoints** in the **Geometric Construction Explorer**.*

10. If needed, create additional geometric constructions.
11. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Coordinate System by Points



Tooltip: CSByPoints

Creates a coordinate system from three points. The first point defines the origin. The second point defines the direction of the first axis. The third point defines the general direction of the second axis.

Required Input	**Output**
Points	Coordinate system
Parameters	
Initial axes to create, such as (x,y) or (y,z)	
Direction of third axis	

Coordinate System by Points Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the

numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the point to use as the coordinate system origin.

2 - Specifies the point to define the direction of the first axis of the value selected for **Assign Axes To**. For example, if **(z,x)** is selected, this point defines the direction of the Z-axis.

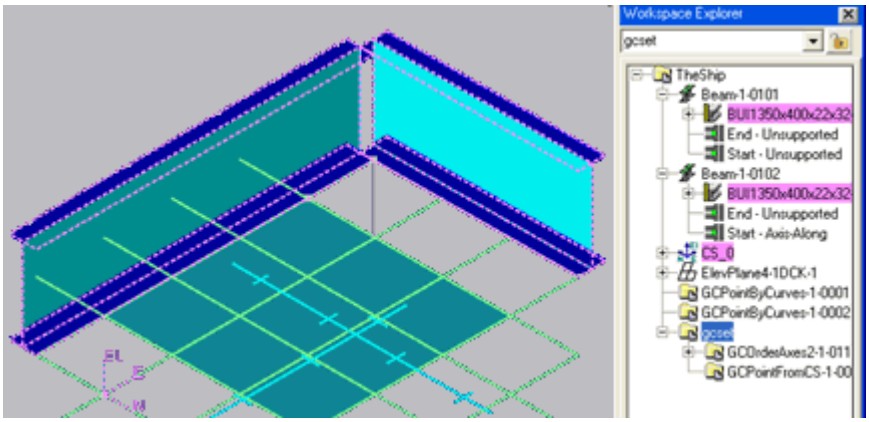
3 - Specifies the point to define the general direction of the second axis of the value selected for **Assign Axes To**. For example, if **(z,x)** is selected, this point defines the general direction of the X-axis. This point must be on the (first axis, second axis) plane.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Assign Axes To - Specifies the axes to define by the selected points. The first value defines the axis for point **2**. The second value defines the axis for point **3**. Select **(x,y)**, **(y,z)**, **(z,x)**, **(y,x)**, **(z,y)**, or **(x,z)**.

Coordinate System - Specifies the direction of the third axis. Select **Direct** to use the right-hand rule to define the third axis. Select **Indirect** to define the third axis in the opposite direction.

What do you want to do?

- *Place a coordinate system at a point* (on page 1175)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)

- *Delete a geometric construction* (on page 1057)

Place a coordinate system at a point

1. Create the points needed to define the coordinate system. Three points are needed for the origin, the direction of the first axis from the origin, and the general direction of the second axis on the (first axis, second axis) plane. Create the points by using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323)

   Points (Geometric Construction Palette Dialog Box) (on page 1028)
2. Click **CSByPoints** on the **Geometric Construction Palette**.
3. In the **Assign Axes To** box, select the initial axes and plane to define.
4. In the **Coordinate System** box, select **Direct**.
5. Select the origin point.
6. Select the point for the direction of the first axis.
7. Select the point for the general direction of the second axis.

   A preview of the coordinate system appears. The direction of the third axis is determined by the right-hand rule.
8. If the third axis is not in the needed direction, select **Indirect** in the **Coordinate System** box.

   The direction of the third axis is flipped.
9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **CSByPoints** in the **Geometric Construction Explorer**.*
10. If needed, create additional geometric constructions.
11. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Solids (Geometric Construction Palette Dialog Box)

The following geometric construction interface creates solids. It is available by selecting **More** in the **Geometric Construction** box of any geometric construction ribbon:

NOTE Each command supporting the 3D option displays tabs and interfaces appropriate to the command.

	Solid by Sweeping - Creates a solid from a set of planar cross-section curves and path curves. For more information, see *Solid by Sweeping* (on page 1176).

Solid by Sweeping

Tooltip: SolidBySweeping

Creates a solid from a set of planar cross-sections and optional path curves. Capping surfaces are created at the ends of the solid when needed.

NOTE This interface is similar to *Surface by Sweeping* (on page 1153).

The cross-section curves:

- Must be planar, but do not need to be coplanar
- Must be oriented in the same direction to avoid a twist in the surface
- Must each have the same number of vertices
- Must be closed

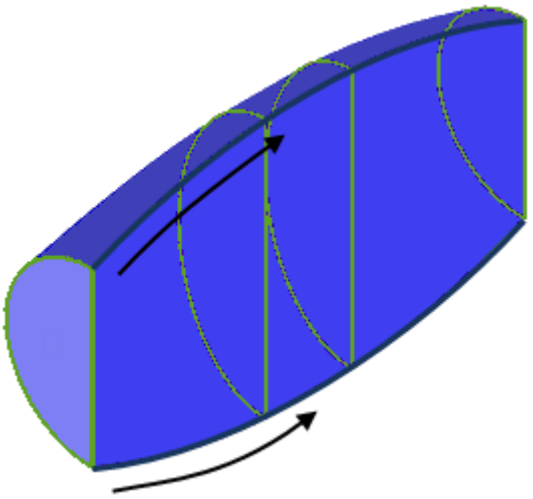
The paths curves:

- Do not need to be planar
- Must touch the cross-section curves unless there is only one path curve
- Must each have the same number of vertices

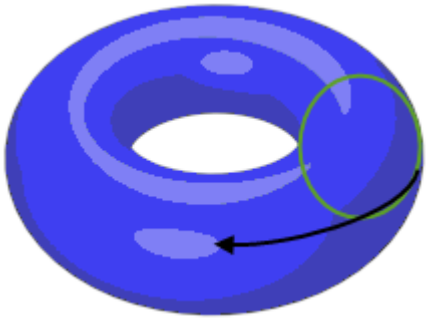
The capping faces are:

- Planar surfaces positioned at and bounded by the first and last cross-section curves
- Merged to the main surface, creating the solid
- Not created if the path curves are closed

The following example shows one planar cross-section curve swept along two open path curves. Capping faces at each end are automatically used to create the closed surfaces of the solid.



The following example shows a torus with a closed cross-section and closed path curve. No capping faces are needed:



Required Input	**Output**
Planar cross-sections	Solid
Optional Input	
Path curves	

Solid by Sweeping Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the cross-section curves.

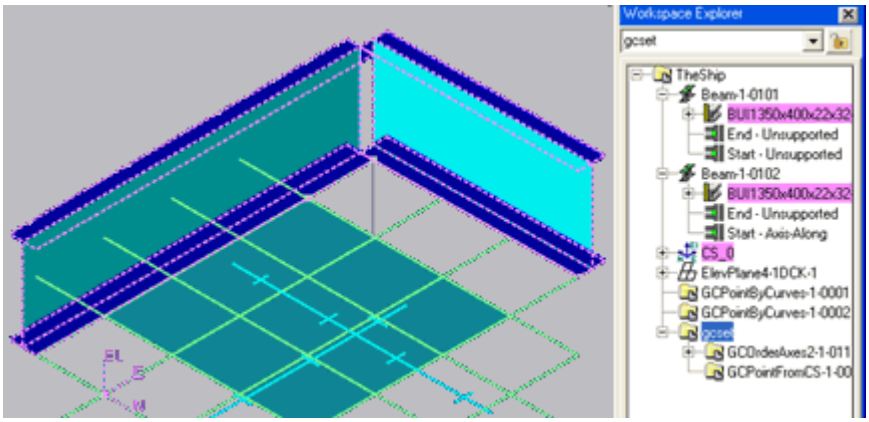
(2) - Specifies one or more path curves. These curves control the creation of the solid. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a solid by sweeping between planar cross-sections* (on page 1180)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a solid by sweeping between planar cross-sections

1. Create the planar cross-section curves needed to define the solid. Optionally, create one or more path curves to control the sweep through the cross-section curves.

   Curves (Geometric Construction Palette Dialog Box) (on page 1061)

2. On the geometric construction ribbon, click **More** in the **Geometric Construction** box.

   *The **Select Geometric Construction** dialog box appears.*

3. Select **SolidBySweeping** and click **OK**.
4. Select one or more cross-section curves in order.
5. Optionally, click (2), and select the path curves.

   A preview of the surface appears.

6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **SolidBySweeping** in the **Geometric Construction Explorer**.*

7. If needed, create additional geometric constructions.
8. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Standard Chutes (Geometric Construction Palette Dialog Box)

You can use chute shape geometric construction macros for quick layout of chute design. The following macros are available to create chute shapes in material handling mode. They appear in the Molded Forms **Advanced Plate System** command on the **Standard Chutes** tab of the **Geometric Construction Palette**.

A complete chute is typically made from three macros. The top macro is the head section, the middle is the transition section, and the bottom is a bifurcated or spoon section.

	Head Section Type A - Creates a chute type A head section. For more information, see Head Section Type A.

	Head Section Type B - Creates a chute type B head section. For more information, see Head Section Type B.
	Head Section Type C - Creates a chute type C head section. For more information, see Head Section Type C.
	Transition Section Type A - Creates a chute type A middle transition section. For more information, see Transition Section Type A.
	Transition Section Type B - Creates a chute type B middle transition section. For more information, see Transition Section Type B.
	Transition Section Type C - Creates a chute type C middle transition section. For more information, see Transition Section Type C.
	Bifurcated Section Type A - Creates a chute type A bottom bifurcated section. For more information, see Bifurcated Section Type A.
	Spoon Section Type A - Creates a chute type A bottom spoon section. For more information, see Spoon Section Type A.

Limitations

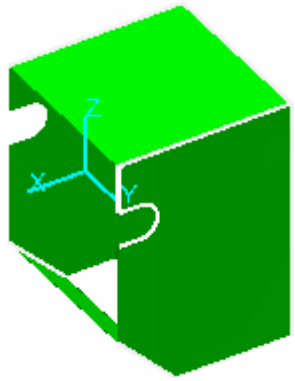
- The software creates the chute as a single plate system with bent knuckles.
- You cannot place stiffener profiles on the chute plate system.
- You cannot place edge reinforcements on the chute plate system.

You can only place openings on the chute plate system by using topological openings. You cannot sketch openings in the Sketch 2D environment. For more information, see *Topological Openings Dialog Box* in the *Molded Forms User's Guide*.

Head Section Type A

Tooltip: HeadSectionTypeA

Creates a chute type A head section.



Required Input	Output
Coordinate system	Chute shape, as a single plate system
Parameters	
Chute dimensions	

Head Section Type A Ribbon

 **Select**

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

 **Add**

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

 **Delete**

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click  again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the coordinate system.

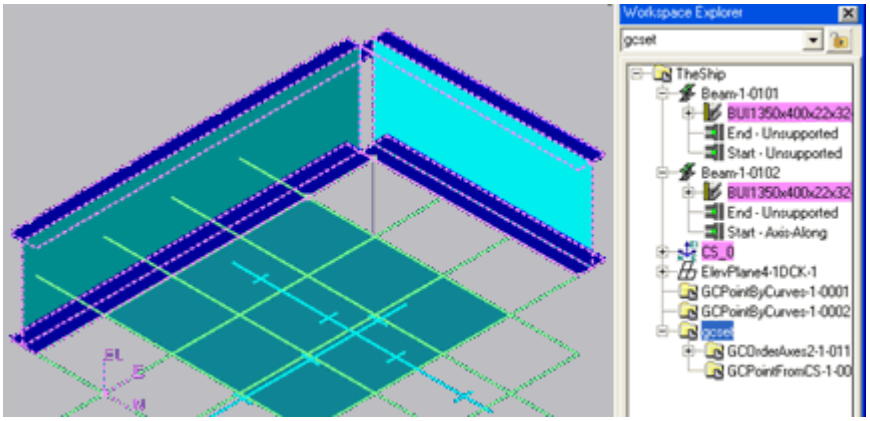
Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background color when there is no input.
-  - Yellow background when an input is selected.

- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

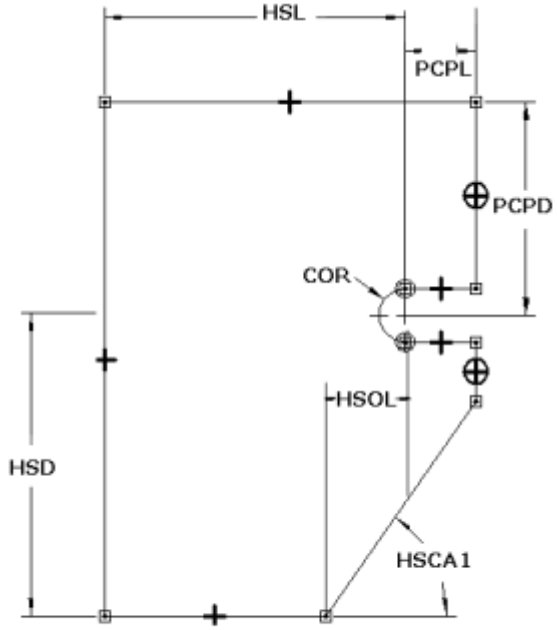
Accepts the current selections and displays a preview.

Continue

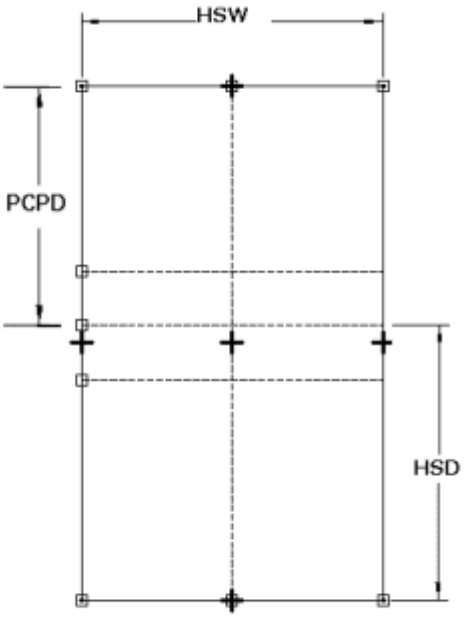
Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Specify the following dimensions:



Elevation



Side Elevation

COR - Cutout radius

PCPD - Pulley centerpoint depth

PCPL - Pulley centerpoint length

HSL - Head section length

HSD - Head section depth

HSW - Head section width

HSOL - Head section opening length

HSCA1 - Head section converging angle

What do you want to do?

- Place a chute shape
- Create a chute using multiple chute shapes
- *Move a chute* (on page 1190)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Place a chute shape

1. In Grids, create a coordinate system at the required inlet location of the chute shape using one of the following commands:
   - **Grid Wizard**
   - **Place Coordinate System**
   - **Place Coordinate System by Three Points**

   For more information, see the *Grids User's Guide.*

   TIP You do not need to create grid planes for the coordinate system.
2. In Molded Forms, click **Advanced Plate System** on the vertical toolbar.

   *The **Advance Plate System** ribbon appears.*
3. Select **Construction** from the **Macro Definition** list.

   OR

   If **Construction** was selected earlier in the same session and is visible in the **Macro Definition** list, click **Plate Definition by 3D**.

   *The **Geometric Construction Palette** dialog box, **Geometric Construction Explorer** dialog box, and **Geometric Construction** ribbon appear.*
4. On the **Standard Chutes** tab of the **Geometric Construction Palette**, click a chute shape macro.

   The ribbon for the macro appears.
5. Select the coordinate system in the graphic view or the **Workspace Explorer**.

   *The chute shape surface name displays in to the **Geometric Construction Explorer**. Green preview geometry displays in the graphic view.*
6. If required, type new parameter values.
7. Click **Continue**.

   The preview geometry color changes to gray in the graphic view.
8. Click **Close**.

   *The **Advance Plate System** ribbon reappears.*
9. Click **Finish**.

The software creates the chute plate system.

Create a chute using multiple chute shapes

Place the head chute shape

1. In Grids, create a coordinate system at the required inlet location of the chute head section using one of the following commands:
   - **Grid Wizard**

- **Place Coordinate System**
- **Place Coordinate System by Three Points**

For more information, see the *Grids User's Guide.*

TIP You do not need to create grid planes for the coordinate system.

2. In Molded Forms, click **Advanced Plate System** on the vertical tool bar.

   *The **Advance Plate System** ribbon appears.*

3. Select **Construction** from the **Macro Definition** list.

   OR

   If **Construction** was selected earlier in the same session and is visible in the **Macro Definition** list, click **Plate Definition by 3D**.

   *The **Geometric Construction Palette** dialog box, **Geometric Construction Explorer** dialog box, and **Geometric Construction** ribbon appear.*

4. On the **Standard Chutes** tab of the **Geometric Construction Palette**, click a head section macro.

   The ribbon for the macro appears.

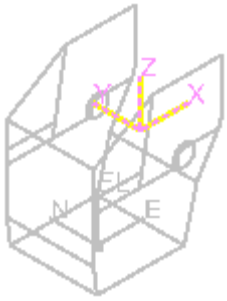
5. Select the coordinate system in the graphic view or the **Workspace Explorer**.

   *The chute shape surface name displays in to the **Geometric Construction Explorer**. Green preview geometry displays in the graphic view.*

6. If needed, type new parameter values.

7. Click **Continue**.

   *The software changes the preview geometry color to gray in the graphic view and adds the head section to the **Geometric Construction Explorer**.*



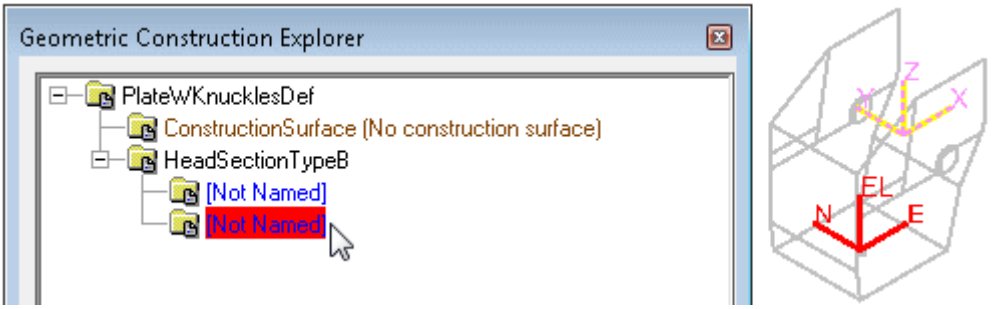
Place the transition chute shape

1. On the **Standard Chutes** tab of the **Geometric Construction Palette**, click a transition section macro.

   The ribbon for the macro appears.

2. Select the local coordinate system at the outlet of the head section. Select the coordinate system in the graphic view or the **Geometric Construction Explorer**.

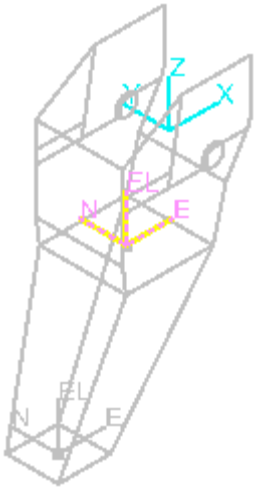
TIP In the **Geometric Construction Explorer**, the coordinate system is one of the two **[Not Named]** items for the head section chute shape.



*The chute shape surface name displays in to the **Geometric Construction Explorer**. Green preview geometry displays in the graphic view.*

3. If required, type new parameter values. Check the compatibility of dimensions where the head section and the transition section meet. For example, if you are using type B sections, the values of **BL** and **W** at the outlet of the head section equal the values of **TSBL** and **TSBW** on the inlet of the transition section.
4. Click **Continue**.

   *The software changes the preview geometry color to gray in the graphic view and adds the transition section to the **Geometric Construction Explorer**.*

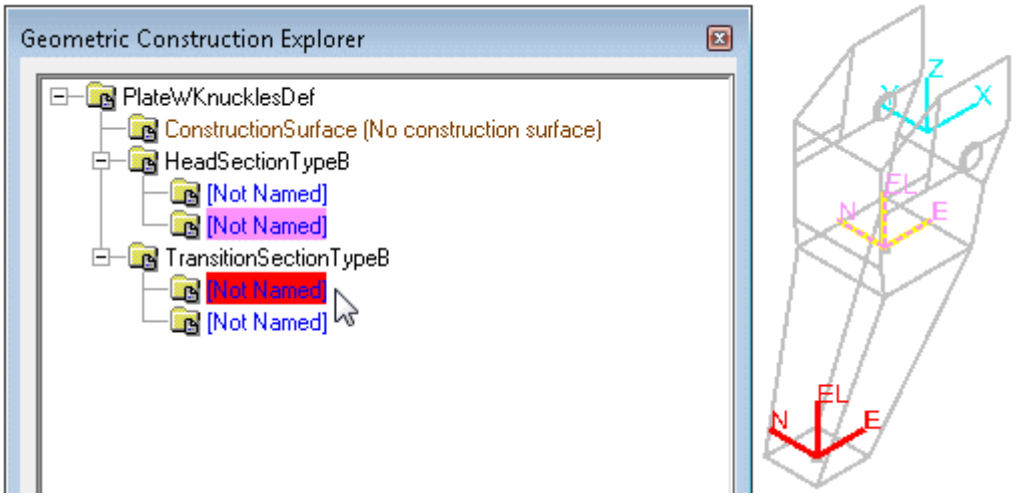


Place the bottom chute shape

1. On the **Standard Chutes** tab of the **Geometric Construction Palette**, click a bottom bifurcated or spoon section macro.

   The ribbon for the macro appears.
2. Select the local coordinate system at the outlet of the transition section. Select the coordinate system in the graphic view or the **Geometric Construction Explorer**.

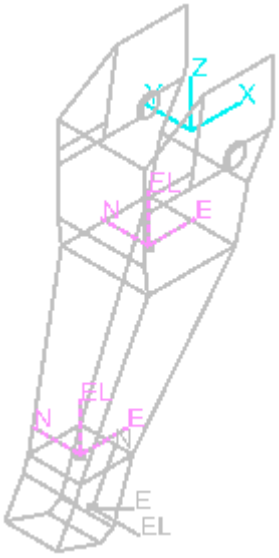
TIP *In the **Geometric Construction Explorer**, the coordinate system is one of the two **[Not Named]** items for the transition section chute shape.*



*The chute shape surface name displays in to the **Geometric Construction Explorer**. Green preview geometry displays in the graphic view.*

3. If required, type new parameter values. Check the compatibility of dimensions where the transition section and the bottom section meet. For example, if you are using type B transition and spoon sections, the values of **TSBOL** and **TSBOW** at the outlet of the transition section equal the values of **SSL**and **SSTW** on the inlet of the spoon section.
4. Click **Continue**.

   *The software changes the preview geometry color to gray in the graphic view and adds the bottom section to the **Geometric Construction Explorer**.*



Merge the chute shapes

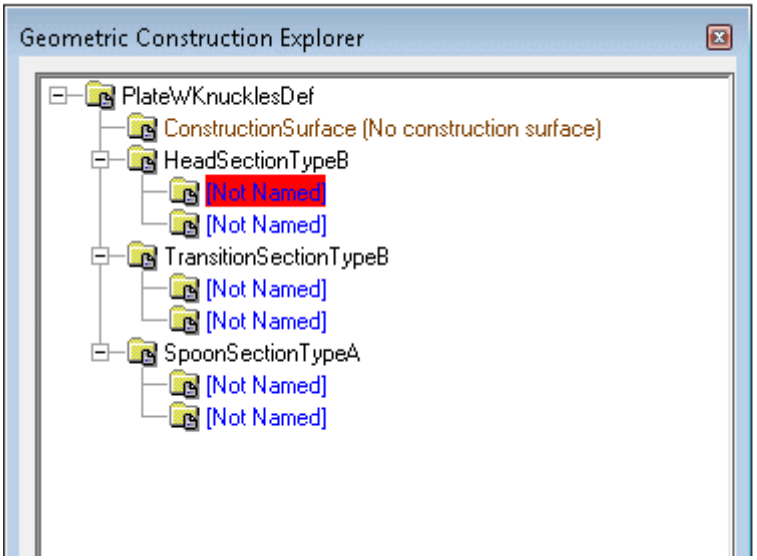
1. On the **Surfaces** tab of the **Geometric Construction Palette**, click **SurfBySurfaces.**

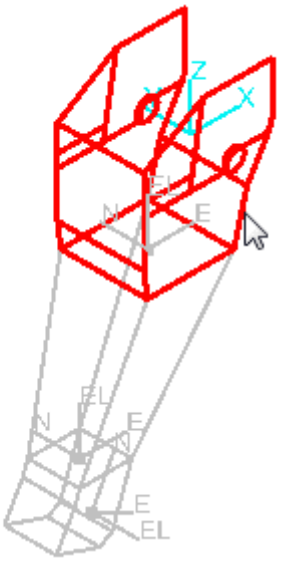


The ribbon for the geometric construction appears.

2. In the **Merge surfaces** box, click **Do not merge co-planar surfaces**.
3. Select each chute shape surface.

TIP You can select the surfaces in the **Geometric Construction Explorer**. For each chute shape, the surface is one of the two **[Not Named]** items.





4. Click **Continue**.

*The software changes the preview geometry color to gray in the graphic view and adds **SurfBySurfaces** to the **Geometric Construction Explorer**.*

5. On the **Surfaces** tab of the **Geometric Construction Palette**, click **SurfWithKnuckles**.



The ribbon for the geometric construction appears.

6. Select **SurfBySurfaces** on the **Geometric Construction Explorer.**

The surface representing the merged chute shapes highlights.

7. Click **Continue**.

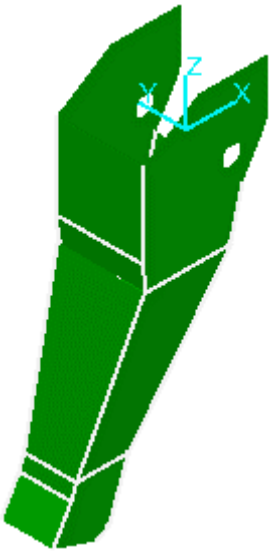
*The software changes the preview geometry color to gray in the graphic view and adds **SurfWithKnuckles** to the **Geometric Construction Explorer**.*

8. Click **Close**.

*The **Advance Plate System** ribbon reappears.*

9. If required, select a parent system and a plate type.
10. Click **Finish**.

    From the merged chute shapes, the software creates a chute as a single plate system with bent knuckles.



Move a chute

1. In the Grids task, click and select the coordinate system used to position a chute.

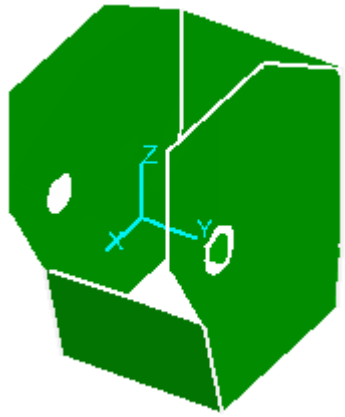
   *The **Place Coordinate System by Three Points** ribbon appears.*
2. Move the coordinate system to the required location.
3. Click **Finish**.

For more information, see *Place Coordinate System by Three Points* in the *Grid User's Guide.*

Head Section Type B

Tooltip: HeadSectionTypeB

Creates a chute type B head section.



Required Input	Output
Coordinate system **Parameters** Chute dimensions	Chute shape, as a single plate system

Head Section Type B Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

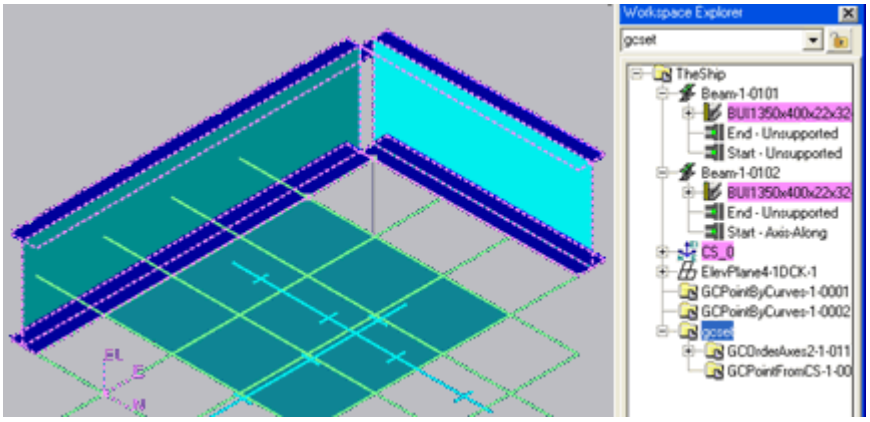
1 - Specifies the coordinate system.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

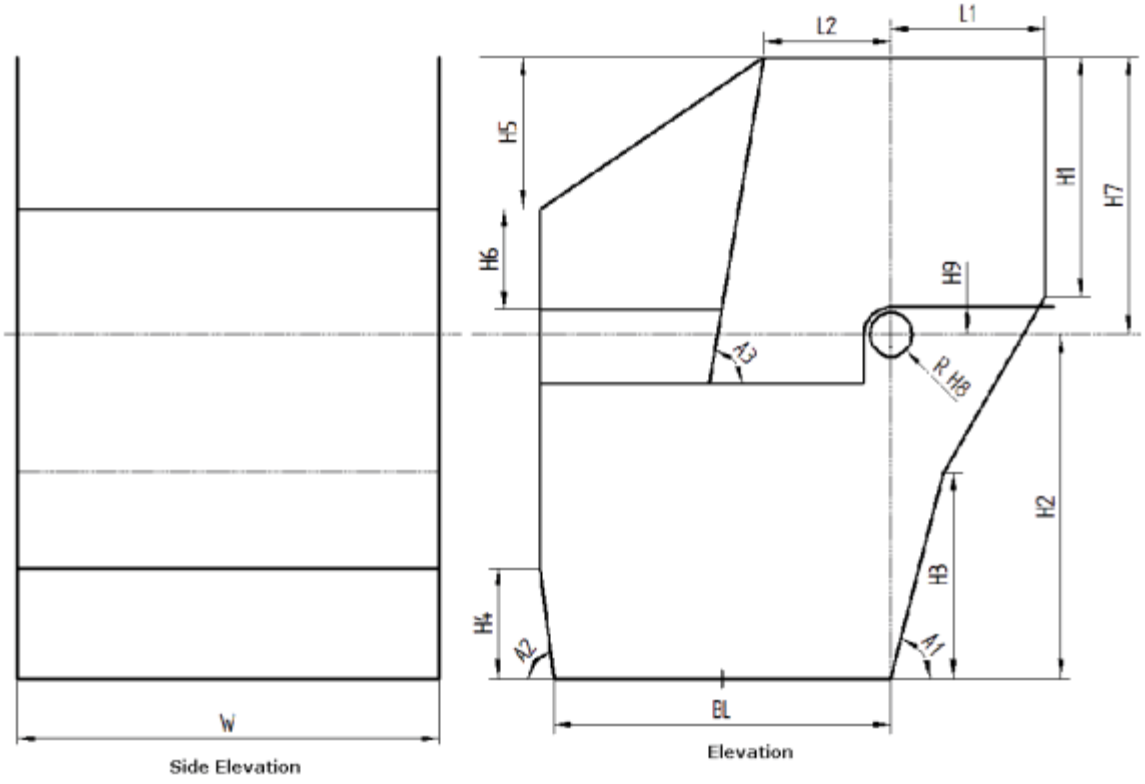
Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Specify the following dimensions:



BL - Head section bottom length

L1, **L2** - Length dimensions

H1, **H2**, **H3**, **H4**, **H5**, **H6**, **H9** - Width dimensions

H7 - Pulley centerpoint depth

H8 - Pulley centerpoint radius

W - Head section width

A1, **A2** - Head section converging angles

A3 - Angle

What do you want to do?

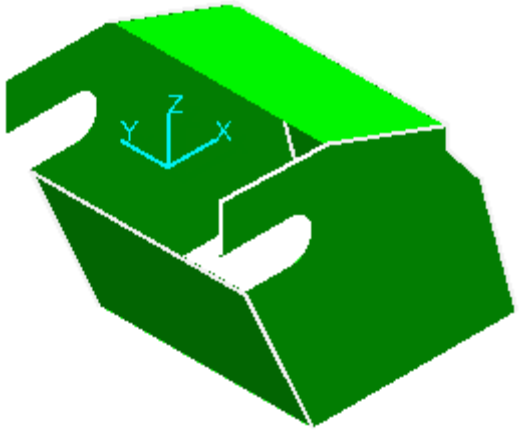
- Place a chute shape
- Create a chute using multiple chute shapes
- *Move a chute* (on page 1190)

- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Head Section Type C

Tooltip: HeadSectionTypeB

Creates a chute type C head section.



Required Input	Output
Coordinate system	Chute shape, as a single plate system
Parameters	
Chute dimensions	

Head Section Type C Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★**IMPORTANT** Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

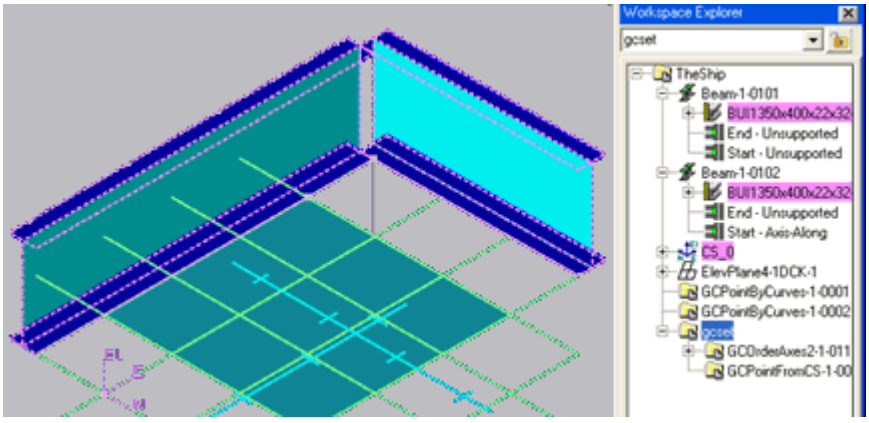
1 - Specifies the coordinate system.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.

- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

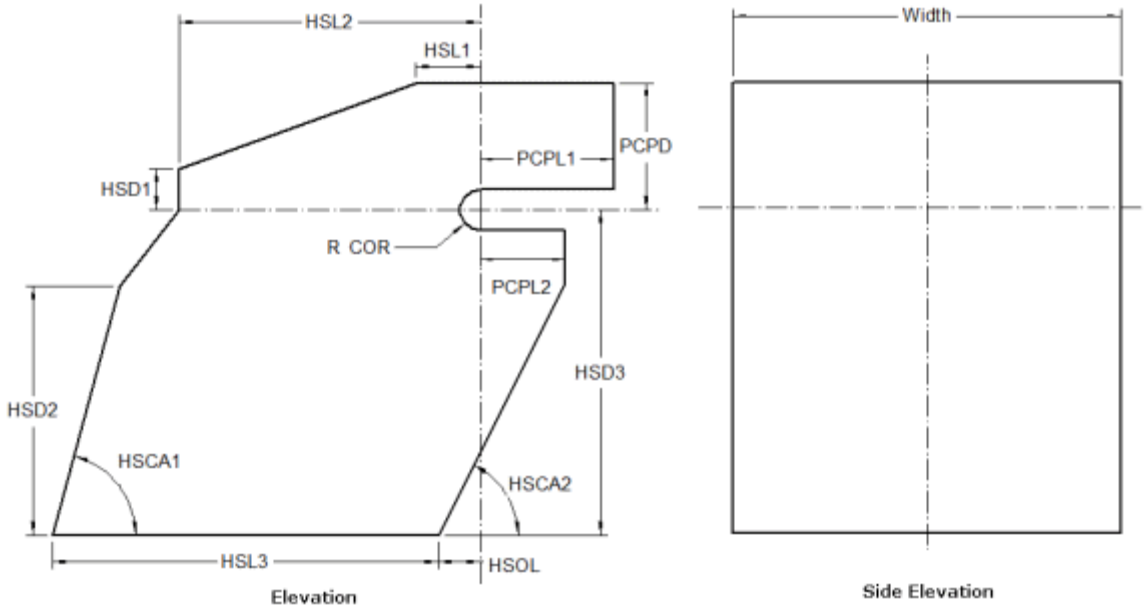
Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Specify the following dimensions:



COR - Cutout radius

Width - Head section width

PCPL1, **PCPL2** - Pulley centerpoint lengths

PCPD - Pulley centerpoint depth

HSL1, **HSL2**, **HSL3** - Head section lengths

HSD1, **HSD2**, **HSD3** - Head section depths

HSW - Head section width

HSOL - Head section opening length

HSCA1, **HSCA2** - Head section converging angles

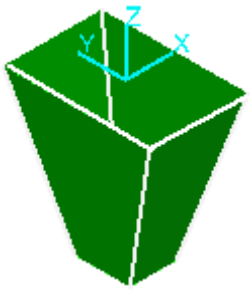
What do you want to do?

- Place a chute shape
- Create a chute using multiple chute shapes
- *Move a chute* (on page 1190)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Transition Section Type A

Tooltip: TransitionSectionTypeA

Creates a chute type A transition section.



Required Input	Output
Coordinate system	Chute shape, as a single plate system
Parameters	
Chute dimensions	

Transition Section Type A Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the coordinate system. When a transition section is the center section of a chute, select the local coordinate system at the outlet of the first (head) section.

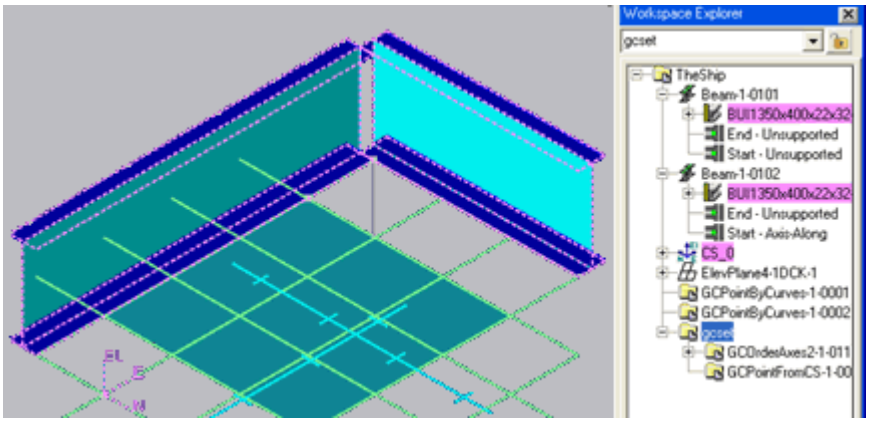


Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

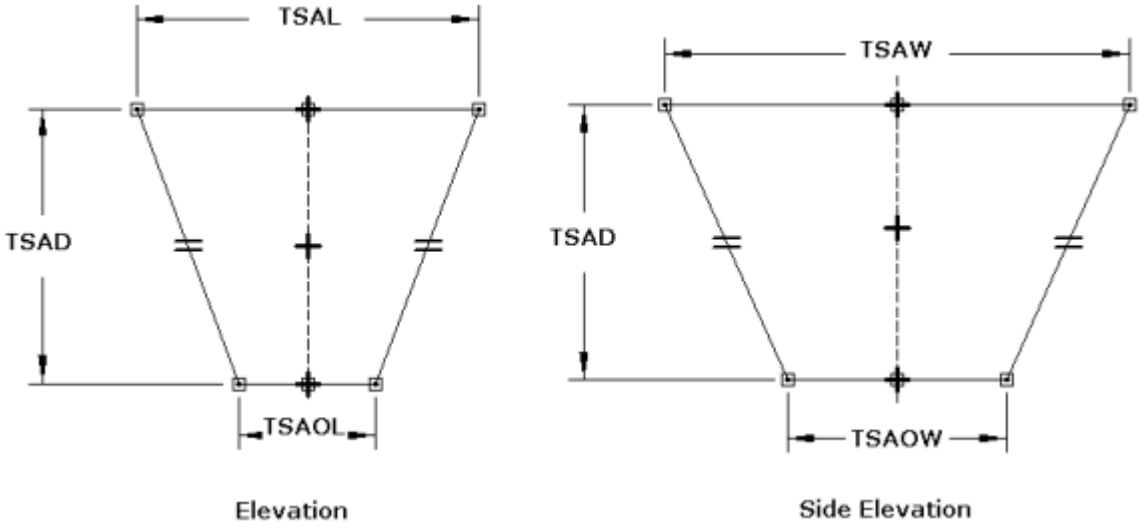
Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Specify the following dimensions:



TSAL - Transition section A length

TSAW - Transition section A width

TSAOL - Transition section A opening length

TSAOW - Transition section A opening width

TSAD - Transition section A depth

TSARA - Transition section A rotation angle of the outlet (TSAOL x TSAOW) relative to the inlet (TSAL x TSAW)

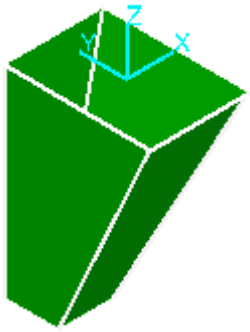
What do you want to do?

- Place a chute shape
- Create a chute using multiple chute shapes
- *Move a chute* (on page 1190)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Transition Section Type B

Tooltip: TransitionSectionTypeB

Creates a chute type B transition section.



Required Input	Output
Coordinate system	Chute shape, as a single plate system
Parameters	
Chute dimensions	

Transition Section Type B Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric**

Construction dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the coordinate system. When a transition section is the center section of a chute, select the local coordinate system at the outlet of the first (head) section.

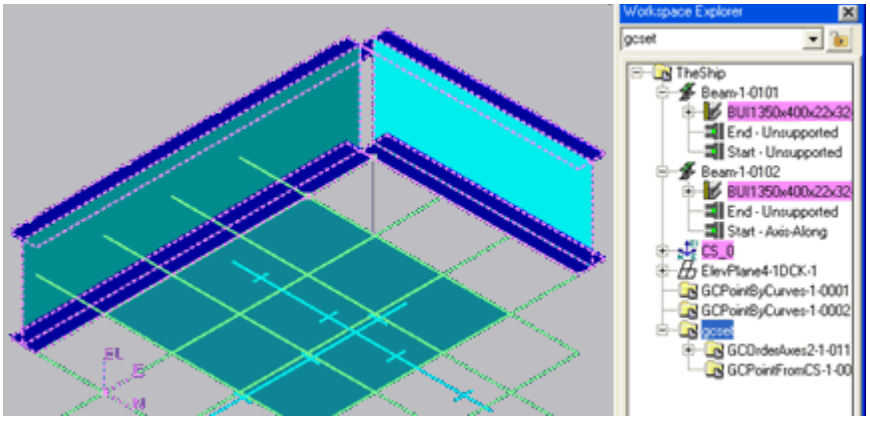


Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

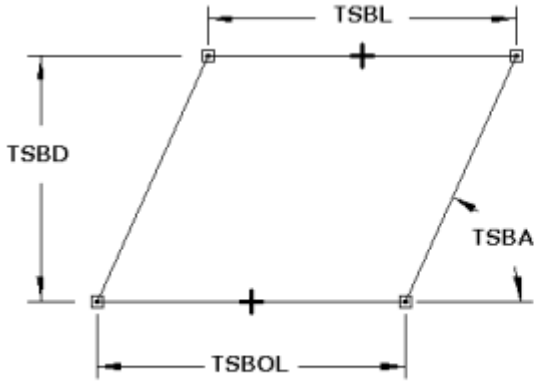
Accepts the current selections and displays a preview.

Continue

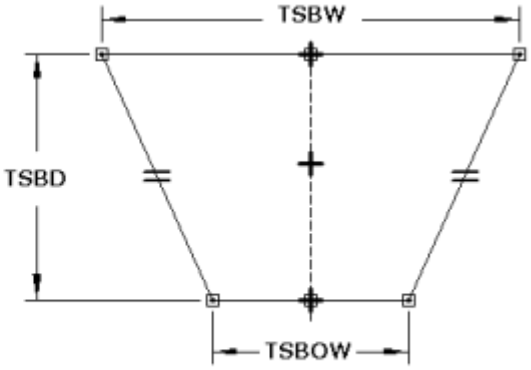
Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Specify the following dimensions:



Elevation



Side Elevation

TSBL - Transition section B length

TSBW - Transition section B width

TSBOL - Transition section B opening length

TSBOW - Transition section B opening width

TSBD - Transition section B depth

TSBA - Transition section B angle

TSBRA - Transition section B rotation angle of the outlet (TBOL x TSBOW) relative to the inlet (TSBL x TSBW)

What do you want to do?

- Place a chute shape
- Create a chute using multiple chute shapes
- *Move a chute* (on page 1190)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Transition Section Type C

Tooltip: TransitionSectionTypeC

Creates a chute type C transition section.



Required Input	Output
Coordinate system	Chute shape, as a single plate system
Parameters	
Chute dimensions	

Transition Section Type C Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the coordinate system. When a transition section is the center section of a chute, select the local coordinate system at the outlet of the first (head) section.

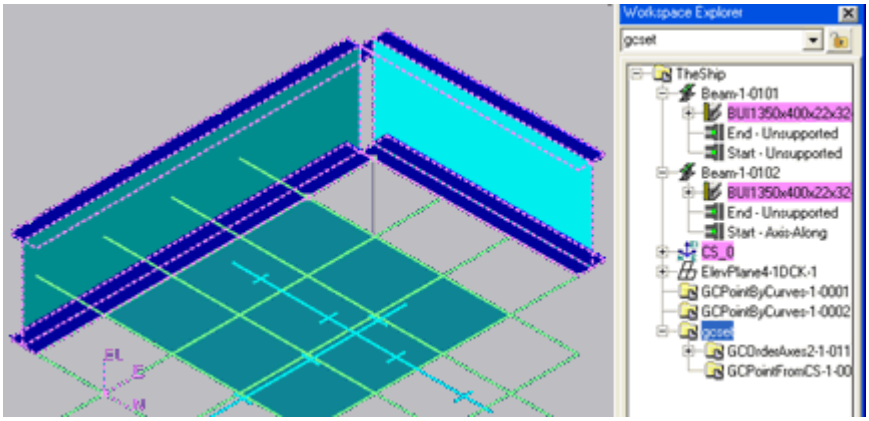


Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.

- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

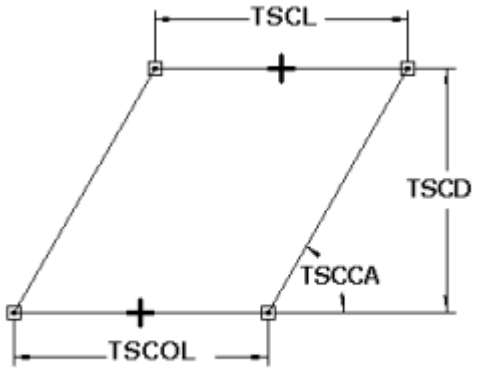
Accepts the current selections and displays a preview.

Continue

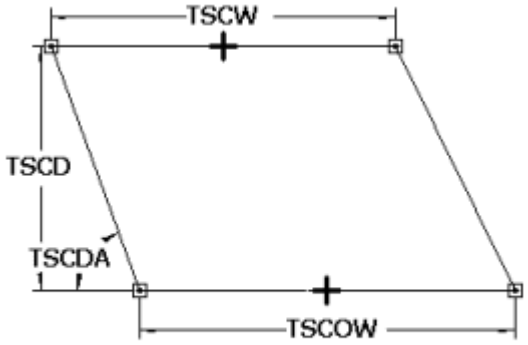
Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Specify the following dimensions:



Elevation



Side Elevation

TSCL - Transition section C length

TSCW - Transition section C width

TSCD - Transition section C depth

TSCOL - Transition section C opening length

TSCOW - Transition section C opening width

TSCCA - Transition section C converging angle

TSCDA - Transition section C diverging angle

TSCRA - Transition section C rotation angle of the outlet (TSCOL x TSCOW) relative to the inlet (TSCL x TSCW)

What do you want to do?

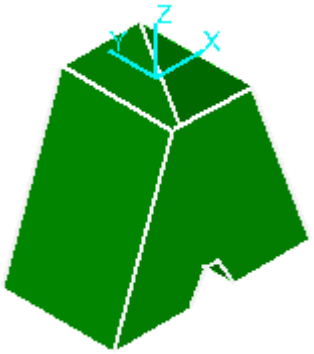
- Place a chute shape
- Create a chute using multiple chute shapes

- *Move a chute* (on page 1190)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Bifurcated Section Type A

Tooltip: BifurcatedSectionTypeA

Creates a chute type A bifurcated section.



Required Input	Output
Coordinate system	Chute shape, as a single plate system
Parameters	
Chute dimensions	

Bifurcated Section Type A Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction

ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

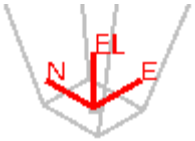
Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the coordinate system. When a bifurcated section is the final section of a chute, select the local coordinate system at the outlet of the center (transition) section.

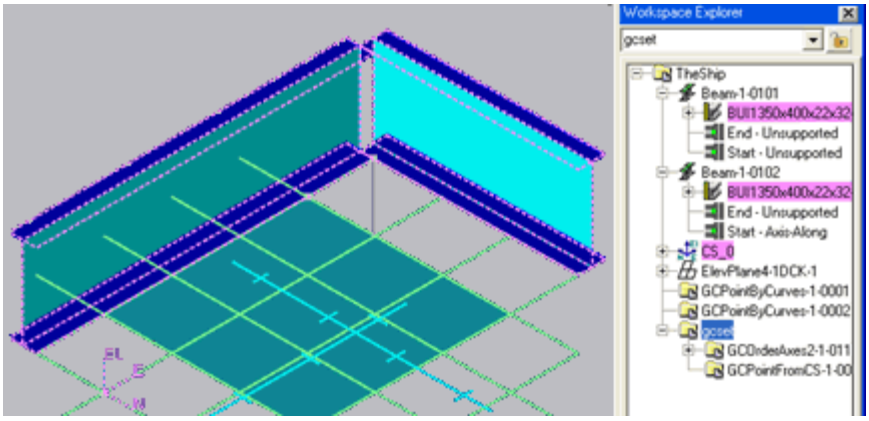


Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

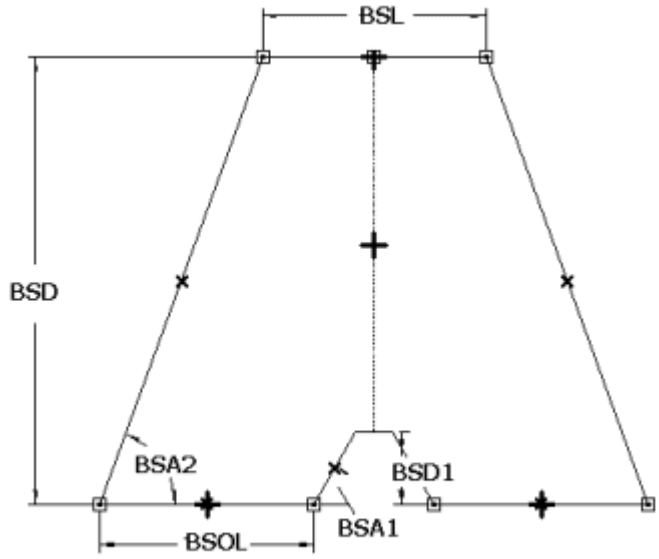
Accepts the current selections and displays a preview.

Continue

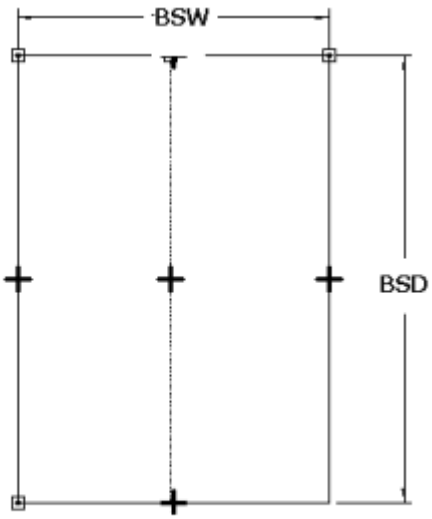
Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Specify the following dimensions:



Elevation



Side Elevation

BSL - Bifurcated section length

BSW - Bifurcated section width

BSOL - Bifurcated section opening length

BSD - Bifurcated section depth

BSD1 - Bifurcated section depth at bifurcation

BSA1 - Bifurcated section angle at bifurcation

BSA1 - Bifurcated section angle

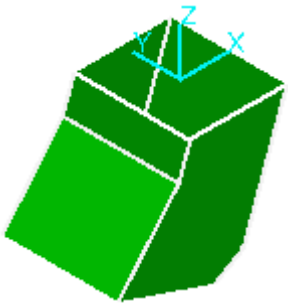
What do you want to do?

- Place a chute shape
- Create a chute using multiple chute shapes
- *Move a chute* (on page 1190)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

Spoon Section Type A

Tooltip: SpoonSectionTypeA

Creates a chute type A spoon section.



Required Input	Output
Coordinate system	Chute shape, as a single plate system
Parameters	
Chute dimensions	

Spoon Section Type A Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

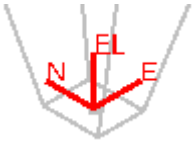
Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the coordinate system. When a spoon section is the final section of a chute, select the local coordinate system at the outlet of the center (transition) section.

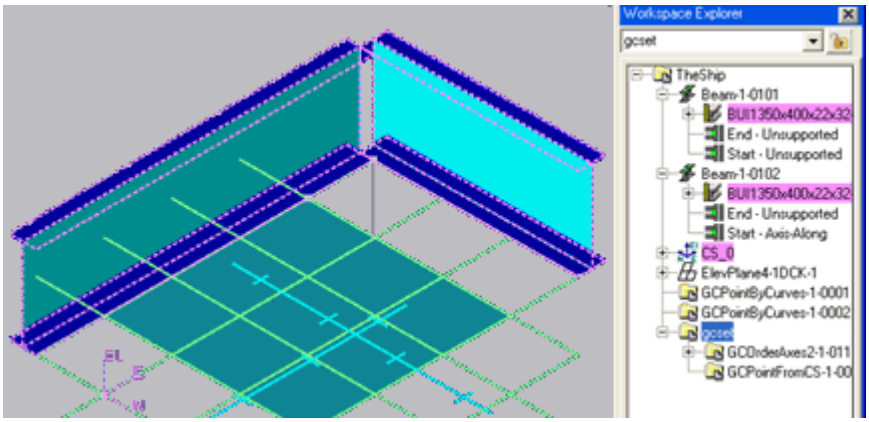


Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

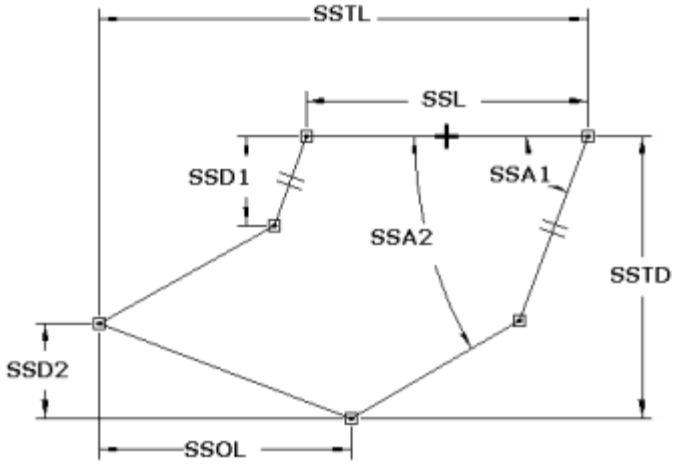
Accepts the current selections and displays a preview.

Continue

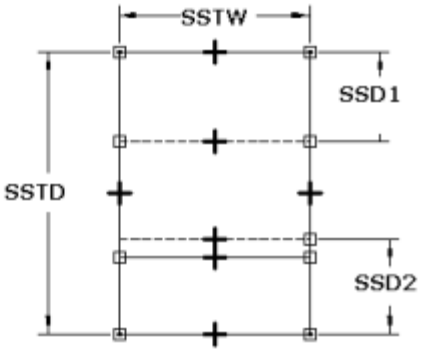
Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Specify the following dimensions:



Elevation



Side Elevation

SSL - Spoon section length

SSTL - Spoon section total length

SSD1 - Spoon section depth 1

SSD2 - Spoon section depth 2

SSOL - Spoon section opening length

SSTD - Spoon section total depth

SSTW - Spoon section total width

SSA1 - Spoon section angle 1

SSA2 - Spoon section angle 2

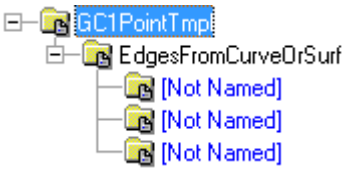
What do you want to do?

- Place a chute shape
- Create a chute using multiple chute shapes
- *Move a chute* (on page 1190)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)

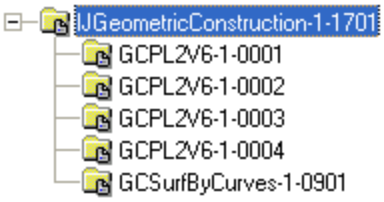
Geometric Construction Explorer

Displays geometry related to the selected geometric construction interface. The geometry is added after you define parameters for the interface and click **Continue** on the **Geometric Construction** ribbon.

When creating a new object, such as a curve or a surface, the geometry is temporary and is not named.



Geometry used to create the object is saved with the object and named. When modifying an existing object, the geometry names display.



The **Geometric Construction Explorer** displays parameters in a hierarchical list similar to the **Workspace Explorer**. Selecting a geometry name highlights the geometry in the graphic view.

Commands

The following commands are available by right-clicking an item in the hierarchical list:

Show

Turns on the graphical display of a hidden object in the graphic view.

Hide

Turns off the graphical display of an object in the graphic view. The names of hidden objects display in italics in the **Geometric Construction Explorer**.

Export

Exports the selected geometric construction to an XML file. This command is only available when you right-click the top level geometric construction set.

Import

Imports a previously exported geometric construction from an XML file into the workspace. The imported geometric construction replaces any existing geometric construction. This command is only available when you right-click the top level geometric construction set.

NOTE **Export** and **Import** are primarily intended for use by administrators creating complex geometric construction macros. The commands can also be used to save multiple geometric constructions, such as a set of seams or profile stiffener landing curves on a hull, that you may not complete in a single session, or that you want to reuse in a similar model.

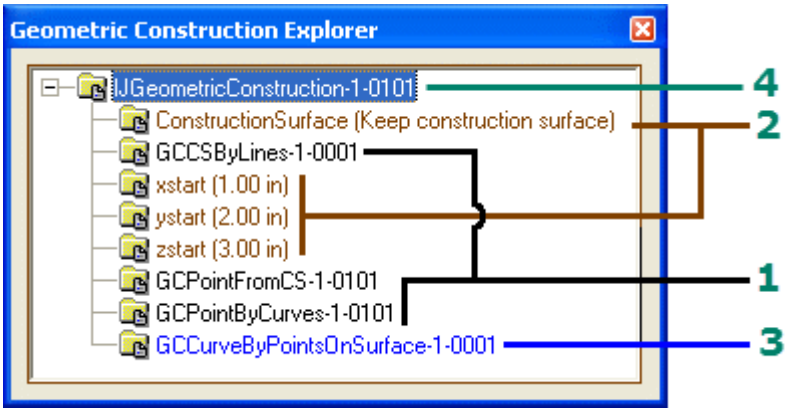
Color Coding

You can identify the purpose of objects by the text color:

Black text - An individual geometric construction macro.

Red text - A shared parameter used by multiple geometric constructions.

Blue text - Output geometry from the macros.



1 - Geometric construction

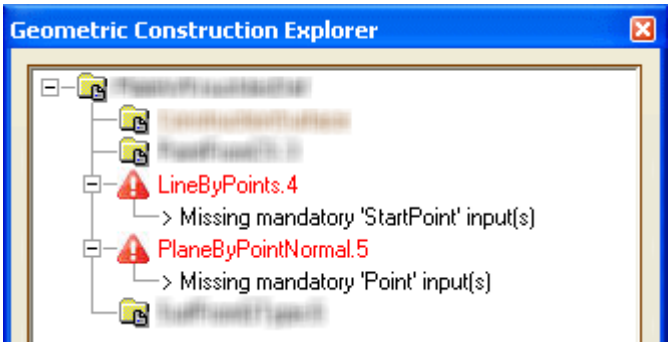
2 - Shared parameter

3 - Output

4 - Geometric construction set

Error Display

When a required input for a geometric construction is deleted, the **Geometric Construction Explorer** displays an error, such as those shown in the following example.



SECTION 18

Importing ACIS and IGES Data

You can temporarily import geometry data in ACIS or IGES format from other software packages, such as Tribon, NAPA, or Rhino. You can compare the temporary geometry to the geometry of objects in the model.

For more information about importing ACIS and IGES data, see *Import ACIS (File Menu)* (on page 1216) and *Import IGES (File Menu)* (on page 1216).

IMPORTANT

- You must have a license for IGES to use the **Import** > **IGES** command.
- The data in the Geometry Analysis and Repair task is not saved in a session file or in the database. If you delete geometry, or define or refresh the workspace, the imported geometry is lost. You must export the data to an external file to save it.
- The imported geometry cannot be used to create objects in the database.

Import ACIS (File Menu)

File > **Import** > **ACIS** imports wireframe and surface data from a file in the Acis .sat format. The imported geometry is not saved in the session file or in the database. After processing the data, you will need to export it to save your work. For more information, see *Export ACIS (File Menu)* (on page 1236).

Import ACIS or IGES data

1. Click **File** > **Import** > **ACIS** or **File** > **Import** > **IGES**, depending on what type of data to import.
2. Browse to locate the file.
3. Click **Open**.

NOTES

- A log file is saved in the folder where the input file resides.
- You can import data over other data. The effect is cumulative.

Import IGES (File Menu)

File > **Import** > **IGES** imports wireframe and surface geometry data from a file in IGES format into the software. The imported geometry is not saved in the session file or in the database. After processing the data, you will need to export it to save your work. For more information, see *Export IGES (File Menu)* (on page 1236).

NOTE You must have a license for IGES in order to use this command.

SECTION 19

Importing Stiffeners and Seams

You can import one or more complete profile stiffener or seam systems from Microsoft Excel workbook files. All stiffener and seam definition types and all data associated with each creation type are supported.

Import modifies existing stiffener or seam objects if objects with the same names already exist on the selected plate system. Relationships and properties of the existing objects are maintained.

Errors are written to a log file if the modification causes incorrect definitions. The software does not overwrite existing log files. It appends new errors to the end of the log file.

Import Seams (File Menu) (on page 1225)
Import Stiffeners (File Menu) (on page 1217)

Import Stiffeners (File Menu)

File > **Import** > **Stiffeners** imports stiffener profile systems from a Microsoft Excel workbook. All stiffener definition types (by intersection, by offset, by projection, and by table) can be imported. All stiffener data associated with each definition type are supported, including names, attributes, cross-section types, wireframe coordinates, boundaries, and orientations.

Import Stiffener Dialog Box (on page 1218)

Stiffener Import Workbook (on page 1218)

Import stiffener or seam data

1. Create an Excel workbook with the stiffener or seam data to be imported. For more information, see *Stiffener Import Workbook* (on page 1218) and *Seam Import Workbook* (on page 1226).
2. Select the root plate system on which you want to import the stiffeners or seams.
3. Click **File > Import > Stiffeners** or **File > Import > Seams**.
4. Define the location of the Excel workbook in **Open File**.
5. Define the location and name of the log file in **Log File**.
6. Click **Submit**.

NOTES

- Click **Cancel** on the **Import Progress** dialog box to stop the import. Processing completes on the current stiffener or seam, updates the log file, and closes the dialog box.
- After import finishes, click **View** to see the log file.

Import Stiffener Dialog Box

Sets options for importing stiffener data.

Plate System

Displays the selected plate system.

Open File

Specifies the name of and path to the Microsoft Excel workbook file containing stiffener data.

- **Browse** - Uses the **Open** dialog box to select the path.
- **View** - Opens the workbook.

Log File

Specifies the name of and path to the log file that is created during import.

- **Browse** - Uses the **Open** dialog box to select the path.
- **View** - Opens the log file.

Submit

Starts importing stiffeners from the specified workbook.

Cancel

Cancels the command and closes the dialog box.

Close

Closes the dialog box. Use after import finishes.

Stiffener Import Workbook

An Excel workbook must have specific formatting to import profile stiffeners. The **stiffener import-export format.xls** workbook is provided as a template of the format needed for each stiffener type. The workbook is delivered to the *[Product Folder]*\MoldedForms\Data folder. Configuration and Summary worksheets in the workbook are provided only for information, and are not required for import.

Each worksheet represents a type of stiffener and the attributes of the stiffener. When a stiffener has been imported into the model, it is like a stiffener of the same type created using the command in the software. Attributes on a worksheet correspond to attributes on the **Properties** dialog box or attributes created using a control on the software ribbon.

Multiple stiffeners of the same type may be defined on each worksheet. Each stiffener definition must end with a **Finish** row. Multiple worksheets may be defined in the same workbook.

Sections common to all worksheets:

Header-Main section

Contains attributes equivalent to using the **Properties** dialog box of the profile ribbon. For more information, see *Main Tab (Profile Properties Dialog Box)* (on page 730).

Name

Name (Main tab)

NamingCategory

Naming Category (Main tab)

Type

Type (Main tab)

Continuity

Continuity (Main tab)

Priority

Continuity Priority (Main tab)

Description

Description (Main tab)

Header-XC section

Contains attributes equivalent to using the **Properties** dialog box of the profile ribbon. For more information, see *Cross Section Tab (Profile Properties Dialog Box)* (on page 732) and *Section Orientation Tab (Profile Properties Dialog Box)* (on page 732).

This section is not used on the Tripping worksheet.

SectionType

Section Type (Cross Section tab)

SectionSize

Section Size (Cross Section tab)

Material

Material (Cross Section tab)

Grade

Grade (Cross Section tab)

PrimaryOrient

Primary Orientation (Section Orientation tab)

SecondaryOrient

Secondary Orientation (Section Orientation tab)

MountingFace

Mounting Face (Section Orientation tab)

LoadPoint

Load Point (Section Orientation tab)

ProjectionMethod

Landing Curve Projection Method (Section Orientation tab)

AttachmentMethod

Attachment Method (Section Orientation tab)

Header-Bound section

Contains attributes equivalent to using the **Boundary Definition** step of the profile ribbon. For more information, see *Boundary Methods* (on page 80).

Boundary 1

Name of the first boundary object. If the boundary is a grid plane, define the name using the grid plane name format (see below).

Boundary 2

Name of the second boundary object. If the boundary is a grid plane, define the name using the grid plane name format (see below).

Ambiguity

Defines whether the boundaries create an ambiguous solution, equivalent to using the **Solve Ambiguity** step of the profile ribbon. Type **True** or **False**. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 82).

TIP The hull is the most common boundary to create an ambiguous solution.

AmbiX

X-coordinate on the selected solution.

AmbiY

Y-coordinate on the selected solution.

AmbiZ

Z-coordinate on the selected solution.

Header-Orient section

Contains attributes equivalent to using the **Profile Orientation Definition** step of the profile ribbon. For more information, see *Profile Orientation* (on page 88).

This section is not used on the Tripping worksheet.

OrientNumber

Sequential number for a point defining rule regions.

PointType

The type of point defining rule regions. Type **StartPoint**, **RegionBoundary**, or **EndPoint**.

IMPORTANT

- The first point must always be **StartPoint**.
- The last point must always be **EndPoint**.

OrientInt.Object

The object intersecting the landing curve to create the definition point. Type **None** or the object name. If the object is a grid plane, define the name using the grid plane name format (see below).

X

X-coordinate of the definition point.

Y

Y-coordinate of the definition point.

Z

Z-coordinate of the definition point.

Rule

Orientation rule used at the definition point. Type **ProfileOrientationNormal**, **ProfileOrientationUpDown**, **ProfileOrientationPortStarboard**, **ProfileOrientationForeAft**, **ProfileOrientationPlanar**, **ProfileOrientationTwistLinearGirth**, **ProfileOrientationTwistLinearOrthogonal**, or **ProfileOrientationTwistCubic**.

Angle

Angle used by the orientation rule at the definition point.

AngleRef

Plane in which to measure the angle. Type **ProfileAnglePlaneNormal** or **ProfileAnglePlaneOrthogonal**.

Plane

Angle reference plane from which the angle is measured. Type **ProfileReferenceNormal**, **ProfileReferenceUpDown**, **ProfileReferencePortStarboard**, or **ProfileReferenceForeAft**.

Intersection worksheet

The section order is: Header-Main, Header-XSec, Header- Bound, Header-Intersect, Header-Orient, and Finish.

Header-Intersection Section

Contains attributes equivalent to using the **Stiffener Profile System by Intersection** command. For more information, see *Stiffener Profile System by Intersection Ribbon* in *Stiffener Profile System by Projection* (on page 746).

Method

Type **Coincident**. This is the required value.

Object

Grid plane used to create the landing curve. Define the name using the grid plane name format (see below).

Offset

Type **0**. This is the required value.

Offset worksheet

The section order is: Header-Main, Header-XSec, Header- Bound, Header-Offset, Header-Orient, and Finish.

Header-Offset section

Contains attributes equivalent to using the **Stiffener Profile System by Offset** command. For more information, see Stiffener Profile System by Offset Ribbon.

BaseCurve

Base curve definition. If the base curve is defined by a grid plane, define the name using the grid plane name format (see below).

Offset

Offset value for the landing curve from the base curve.

OffsetMethod

Method used to offset the landing curve from the base curve. Type **Girth**, **Vertical**, **Transverse**, or **Longitudinal**.

Projection worksheet

The section order is: Header-Main, Header-XSec, Header- Bound, Header-Project, Header-Orient, and Finish.

Header-Project section

Contains attributes equivalent to using the **Stiffener Profile System by 2D Projection** command. For more information, see Stiffener Profile System by 2D Projection Ribbon.

SketchPlaneMethod

Sketch plane method used. Type **Coincident** or **Offset**. Other sketch plane methods, such as Angle from Plane, Plane by Point & Vector, Plane by Three Points, and Plane by Vectors Normal, cannot be used for import.

Offset

Offset value for the sketch plane if the **Offset** SketchPlaneMethod type is used. Type **0** if the **Coincident** SketchPlaneMethod type is used.

Object

Grid plane or object used to create as the sketch plane. If the object is a grid plane, define the name using the grid plane name format (see below).

FileName

Type the fully qualified path and the .sha file name containing the sketch curve. If the .sha file is in the same folder as the import workbook, then only the file name is required.

Table worksheet

The section order is: Header-Main, Header-XSec, Header- Bound, Header-Table, Header-IntPlane, Header-Orient, and Finish.

Header-Table section

Contains attributes equivalent to using the **Stiffener Profile System by Table** command. For more information, see *Stiffener Profile System by Table Ribbon* in *Stiffener Profile System by Table* (on page 755) and *Landing Curve Table Dialog Box* (on page 760).

LCNumber

Sequential number for a point on the landing curve.

PointType

Type **Definition** or **Knuckle**.

IntTab.Object

Intersecting object that helps to define two coordinate values of the landing curve point.

X-Reference

X-coordinate of the landing curve point. Type **0** if the coordinate is derived.

X-Offset

Offset value form the X-reference. Type **0** is there is no offset.

Y-Reference

Y-coordinate of the landing curve point. Type **0** if the coordinate is derived.

Y-Offset

Offset value form the Y-reference. Type **0** is there is no offset.

Z-Reference

Z-coordinate of the landing curve point. Type **0** if the coordinate is derived.

Z-Offset

Offset value form the Z-reference. Type **0** is there is no offset.

DerivedCoord

Type **X**, **Y**, or **Z** for the derived coordinate.

Header-IntPlane section

Contains attributes equivalent to using the **Intersection Plane** option of the **Stiffener Profile System by Table**  command. This section is optional and is used when the coordinates of only two points are known for a landing curve on a plate system with curvature. For more information, see *Stiffener Profile System by Table Ribbon* in *Stiffener Profile System by Table* (on page 755) and *Landing Curve Table Dialog Box* (on page 760).

BasePlane

Grid plane *in* which to measure the angle used to define the intersection plane. Define the name using the grid plane name format (see below).

NormalToPlane

Normal to Base Plane. Defines whether the intersection plane is oriented normal to the base plane. Type **True** or **False**.

MeasureFrom

Defines the plane *from* which to measure the angle used to define the intersection plane. Type **Horizontal** or **Vertical**.

IntPlaneAngle

Angle. The angle of the intersection plane, measured from the **Horizontal** or **Vertical** plane.

Tripping worksheet

The section order is: Header-Main, Header-XSec, Header- Trip, and Finish.

Header-Trip section

Contains attributes equivalent to using the **Tripping Stiffener** command. For more information, see *Tripping Stiffener System Ribbon* in *Tripping Stiffener System* (on page 765).

B1

Starting object for the tripping stiffener.

B1ConnectType

Connection type between the tripping stiffener and starting object. Type **Connected**, **Lapped**, **Parallel**, **Perpendicular**, **Transverse**, **Vertical**, **Longitudinal**, **Fixed X**, **Fixed Y**, **Fixed Z**, or **End Connect**.

B2

Ending object for the tripping stiffener.

B2ConnectType

Connection type between the tripping stiffener and ending object. Type **Connected**, **Lapped**, **Parallel**, **Perpendicular**, **Transverse**, **Vertical**, **Longitudinal**, **Fixed X**, **Fixed Y**, **Fixed Z**, or **End Connect**.

Configuration worksheet

Date

The date on which the export was performed. This field can be blank for import.

IntelliShip Version

The version of software from which the export was performed. This field can be blank for import.

User

The name of the active user. This field can be blank for import.

ModelDB

The name of the model database. This field can be blank for import.

CatalogDB

The name of the catalog database. This field can be blank for import.

PlateSystem

The plate system name. This field can be blank for import.

Number Exported

The number of stiffeners included in the export. This field can be blank for import.

CoordSystemSepa

A coordinate system separator character that is not part of the coordinate system or reference plane names.

Use the same character throughout the workbook. "@" is recommended.

Do not use the backslash character. ("****").

Distance Unit

This unit can be meters (**M**) or millimeters (**MM**).

Angle Unit

This unit can be **Degrees** or **Radians**.

Grid Plane Name Format

If an object is a grid plane, define the name by typing the coordinate system, a coordinate system separator character, and the grid plane name. For example, *CS_O@L-21*.

Choose a coordinate system separator character that is not part of the coordinate system or reference plane names.

Use the same character throughout the workbook. "@" is recommended.

Do not use the backslash character. ("****").

See Also

Import Stiffener or Seam Data (on page 1217)
Import Stiffeners (File Menu) (on page 1217)

Import Seams (File Menu)

File > **Import** > **Seams** imports design seams from a Microsoft Excel workbook. All seam definition types (by intersection, by offset, by projection, and by table) can be imported. All seam data associated with each definition type are supported, including names, attributes, wireframe coordinates, and boundaries.

Import Seam Dialog Box (on page 1225)

Seam Import Workbook (on page 1226)

What do you want to do?

- *Import Stiffener or Seam Data* (on page 1217)

Import Seam Dialog Box

Sets options for importing seam data.

Plate System

Displays the selected plate system.

Open File

Specifies the name of and path to the Microsoft Excel workbook file containing seam data.

- **Browse** - Uses the **Open** dialog box to select the path.
- **View** - Opens the workbook.

Log File

Specifies the name of and path to the log file that is created during import.

- **Browse** - Uses the **Open** dialog box to select the path.
- **View** - Opens the log file.

Submit

Starts importing seams from the specified workbook.

Cancel

Cancels the command and closes the dialog box.

Close

Closes the dialog box. Use after import finishes.

Seam Import Workbook

An Excel workbook must have specific formatting to import design seams. The **stiffener import-export format.xls** workbook is provided as a template of the format needed for each seam type. The workbook is delivered to the *[Product Folder]*\MoldedForms\Data folder. Configuration and Summary worksheets in the workbook are provided only for information, and are not required for import.

Each worksheet represents a type of seam and the attributes of the seam. When a seam has been imported into the model, it is like a seam of the same type created using the command in the software. Attributes on a worksheet correspond to attributes on the **Properties** dialog box or attributes created using a control on the software ribbon.

Multiple seams of the same type may be defined on each worksheet. Each seam definition must end with a **Finish** row. Multiple worksheets may be defined in the same workbook.

Sections common to all worksheets:

Header-main section

Contains attributes equivalent to using the **Properties** dialog box of the profile ribbon. For more information, see *Main Tab (Seam Properties Dialog Box)* (on page 904).

Name

Name (Main tab)

Type

Type (Main tab). Type **DesignSeam**.

Header-Bound section

Contains attributes equivalent to using the **Boundary Definition** step of the profile ribbon. For more information, see *Boundary Methods* (on page 80).

Boundary 1

Name of the first boundary object. If the boundary is a grid plane, define the name using the grid plane name format, below.

Boundary 2

Name of the second boundary object. If the boundary is a grid plane, define the name using the grid plane name format, below.

Ambiguity

Defines whether the boundaries create an ambiguous solution, equivalent to using the **Solve Ambiguity** step of the profile ribbon. Type **True** or **False**. For more information, see

Solve ambiguous solution created by selected boundaries (on page 82).

TIP The hull is the most common boundary to create an ambiguous solution.

AmbiX

X-coordinate on the selected solution.

AmbiY

Y-coordinate on the selected solution.

AmbiZ

Z-coordinate on the selected solution.

Intersection worksheet:

The section order is: Header-Main, Header-Bound, Header- Intersect, and Finish.

Header-Intersect section

Contains attributes equivalent to using the **Seam by Intersection** command. For more information, see *Seam by Intersection Ribbon* in *Seam by Intersection* (on page 902).

Method

Type **Coincident**. This is the required value.

Object

Grid plane used to create the landing curve. Define the name using the grid plane name format, below.

Offset

Type **0**. This is the required value.

Offset worksheet:

The section order is: Header-Main, Header-Bound, Header- Offset, and Finish.

Header-Offset section

Contains attributes equivalent to using the **Seam by Offset** command. For more information, see *Seam by Offset Ribbon* in *Seam by Offset* (on page 905).

BaseCurve

Base curve definition. If the base curve is defined by a grid plane, define the name using the grid plane name format, below.

Offset

Offset value for the landing curve from the base curve.

OffsetMethod

Method used to offset the landing curve from the base curve. Type **Girth**, **Vertical**, **Transverse**, or **Longitudinal**.

Projection worksheet:

The section order is: Header-Main, Header-Bound, Header- Project, and Finish.

Header-Project section

Contains attributes equivalent to using the **Design Seam by Projection** command. For more information, see *Seam by Projection* (on page 908).

SketchPlaneMethod

Sketch plane method used. Type **Coincident** or **Offset**. Other sketch plane methods, such as Angle from Plane, Plane by Point & Vector, Plane by Three Points, and Plane by Vectors Normal, cannot be used for import.

Offset

Offset value for the sketch plane if the **Offset** SketchPlaneMethod type is used. Type **0** if the **Coincident** SketchPlaneMethod type is used.

Object

Grid plane or object used to create as the sketch plane. If the object is a grid plane, define the name using the grid plane name format, below.

FileName

Type the fully qualified path and the .sha file name containing the sketch curve. If the .sha file is in the same folder as the import workbook, then only the file name is required.

Table worksheet:

The section order is: Header-Main, Header-Bound, Header- Table, Header-IntPlane, and Finish.

Header-Table section

Contains attributes equivalent to using the **Stiffener Profile System by Table** command. For more information, see *Stiffener Profile System by Table Ribbon* in *Stiffener Profile System by Table* (on page 755) and *Landing Curve Table Dialog Box* (on page 760).

LCNumber

Sequential number for a point on the landing curve.

PointType

Type **Definition** or **Knuckle**.

IntTab.Object

Intersecting object that helps to define two coordinate values of the landing curve point.

X-Reference

X-coordinate of the landing curve point. Type **0** if the coordinate is derived.

X-Offset

Offset value form the X-reference. Type **0** is there is no offset.

Y-Reference

Y-coordinate of the landing curve point. Type **0** if the coordinate is derived.

Y-Offset

Offset value form the Y-reference. Type **0** is there is no offset.

Z-Reference

Z-coordinate of the landing curve point. Type **0** if the coordinate is derived.

Z-Offset

Offset value form the Z-reference. Type **0** is there is no offset.

DerivedCoord

Type **X**, **Y**, or **Z** for the derived coordinate.

Header-IntPlane section

Contains attributes equivalent to using the **Intersection Plane** option of the **Stiffener Profile System by Table** command. This section is optional and is used when the coordinates of only two points are known for a landing curve on a plate system with curvature. For more information, see *Stiffener Profile System by Table Ribbon* in *Stiffener Profile System by Table* (on page 755) and *Landing Curve Table Dialog Box* (on page 760).

BasePlane

Grid plane *in* which to measure the angle used to define the intersection plane. Define the name using the grid plane name format, below.

NormalToPlane

Normal to Base Plane. Defines whether the intersection plane is oriented normal to the base plane. Type **True** or **False**.

MeasureFrom

Defines the plane *from* which to measure the angle used to define the intersection plane. Type **Horizontal** or **Vertical**.

IntPlaneAngle

Angle. The angle of the intersection plane, measured from the **Horizontal** or **Vertical** plane.

Configuration worksheet:

Date

The date on which the export was performed. This field can be blank for import.

IntelliShip Version

The version of software from which the export was performed. This field can be blank for import.

User

The name of the active user. This field can be blank for import.

ModelDB

The name of the model database. This field can be blank for import.

CatalogDB

The name of the catalog database. This field can be blank for import.

PlateSystem

The plate system name. This field can be blank for import.

Number Exported

The number of stiffeners included in the export. This field can be blank for import.

CoordSystemSepa

A coordinate system separator character that is not part of the coordinate system or reference plane names.

Use the same character throughout the workbook. "@" is recommended.

Do not use the backslash character. ("****").

Distance Unit

This unit can be meters (**M**) or millimeters (**MM**).

Angle Unit

This unit can be **Degrees** or **Radians**.

Grid Plane Name Format

If an object is a grid plane, define the name by typing the coordinate system, a coordinate system separator character, and the grid plane name. For example, *CS_O@L-21*.

Choose a coordinate system separator character that is not part of the coordinate system or reference plane names.

Use the same character throughout the workbook. "@" is recommended.

Do not use the backslash character. ("****").

See Also

Import Seams (File Menu) (on page 1225)
Import Stiffener or Seam Data (on page 1217)

SECTION 20

Import Ship Structure

File > **Import** > **Ship Structure**, available in the Molded Forms and Structural Detailing tasks, imports Tribon marine structure XML data into Smart 3D for planning and outfitting work.

To use this command, you must purchase and install the **Intergraph Smart™ 3D Tribon Interface** (SEBY812AG) from Smart 3D. Please contact Intergraph Support for licensing information.

Before you can import the data into Smart 3D, you must export the data from Tribon Initial Design Version M3 (Version 1.3). Smart 3D only supports XML in Schema V3 format.

***Import Ship Structure Dialog Box* (on page 1231)**

Import ship structure data

1. Export data from Tribon Initial Design to create an M3 XML file.
2. In Smart 3D, select **File** > **Import** > **Ship Structure**.

   *The **Import Ship Structure** dialog box appears.*
3. Click the button next to the **XML file or path** box to navigate to the appropriate folder location, and do one of the following:

   Click **Open** to process all of the valid XML files in the folder.

   OR

   Select the individual XML file to use for import, and then click **Open**.
4. Click the button next to the **Map file** box to select the XLS map workbook to use for import, and then click **Open**.
5. Select a system to use as the parent for the imported data.
6. Click **OK** to import the Tribon data as specified, or click **Submit Job** to run the import as a batch job.

   NOTE For more information about setting up the batch service, see *Installing Batch Services* in the *Intergraph Smart™ 3D Installation Guide*.

Import Ship Structure Dialog Box

Sets options for importing Tribon XML data.

XML file or path

Specify the name of and path to the XML file exported from Tribon. To process multiple files simultaneously, specify the name of and path to the folder that contains the XML files to import. Alternatively, you can use the browse feature and navigate to the file or folder of files to be used for import.

Map file

Specify the name of and path to the mapping file between the Tribon XML data and the Smart 3D data in the catalog. For more information, see *Import Translation Map Workbook* (on page 1232).

System

Select a system in the Smart 3D model under which to place the imported data.

Import Translation Map Workbook

The Microsoft Excel workbook named **SM_Mapping.xls** is delivered in the *[Reference Data Folder]*\SharedContent\Data\TribonTranslator\MappingFiles folder. This workbook maps Tribon material and cross-section names to Smart 3D material and cross-section names.

The Cross Section worksheet maps profile cross-section names between the Tribon XML file and Smart 3D.

Cross Section Worksheet (on page 1232)

The Material worksheet maps Tribon material names to Smart 3D material names.

Material Worksheet (on page 1232)

Cross Section Worksheet

The **Cross Section** sheet of the **SM_Mapping.xls** workbook maps Tribon profile cross-section names in the XML file to Smart 3D profile cross-section names.

Section Name

Type the Smart 3D profile cross-section name.

Imported Family Name

Type the Tribon profile cross-section name as it appears in the XML file.

NOTE The last item listed for each section type is the default value for that section type.

Material Worksheet

The **Material** sheet of the **SM_Mapping.xls** workbook maps Tribon materials in the XML file to Smart 3D materials.

Material Type

Type the Smart 3D material type.

Material Grade

Type the Smart 3D material grade.

Imported Material Grade

Type the Tribon material name as it appears in the XML file.

SECTION 21

Import Bocad

The **File > Import > Structure > Bocad** command imports a Bocad® .bti file into Smart 3D.

Import of Attributes/Properties

Attributes that were defined in bocad and exported can be mapped and imported to Smart 3D properties using the sample UserAttribute_Map.xlm file. These attributes are included in the BOCAD_Master_Physical_Map.xml file. This file is delivered with Smart 3D in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

If the bocad attribute that you want to map does not have an equivalent property in Smart 3D, you need to add that property to Smart 3D (using bulkload) before you can map to it in the xml file.

If the bocad attribute does not have an entry in the mapping files but a property with the same name exists in the Smart 3D, then that third-party attribute imports and populates the Smart 3D property even though it is not in the mapping file.

Member Orientation and Member Type

The physical orientation of the member determines the member type during import.

- If the bocad member name exactly matches an existing member type defined in Smart 3D, then the software sets the member to that member type regardless of orientation. For example, if a member has an orientation greater than 10° from vertical but is named "Column", the software imports that member as a column despite the member having an orientation of a vertical brace.
- Members that are vertical or within 10° (1/10 slope) of vertical import as columns.
- Members that are horizontal or within 10° of horizontal import as beams.
- Members that are between 10° of horizontal and 10° of vertical import as vertical braces.
- Members are never imported as horizontal braces as there is no way to distinguish horizontal braces from beams. You must manually edit the members that are horizontal braces after import and set the correct member type.
- All curved members import as beams.

Limitations

- Curved members are imported using the resolution defined in the import file.
- Built-up members are imported if the mapping file is defined to do that.
- Previously imported structure is not updated on subsequent imports. In this situation, either that structure is skipped, replaced, or imported as a copy depending on the import option chosen.

How to Import a bocad File

1. Select **File > Import > Structure > Bocad**.

2. Select the system and coordinate system to use for the imported objects.
3. Select whether to import members (with or without openings) and plates (with or without openings).
4. Define the .bti file to import.
5. Select the mapping file.
6. Define the location and name for the log file.
7. Select the action to perform when you click **Import**.

Click **Import**.

bocad Import Dialog Box

Controls how objects are imported from the bocad .bti file. Be sure to check the Compatibility Matrix before you import data. Open *https://smartsupport.intergraph.com* *https://smartsupport.intergraph.com*, select **View Downloads** at the top, and then select **Product Compatibility** under **Useful Links** on the right.

System

Select the system to which to assign the objects being imported. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model.

Coordinate System

Select the coordinate system to use for the imported objects. All objects are placed relative to this coordinate system.

Members

Select to import members in the import file. Clear this option to not import members in the import file.

Plates

Select to import plates in the import file. Clear this option to not import plates in the import file.

Openings

Select to import opening in members and plates. Clear this option to not import openings.

Import File

Define the file name and folder path of the bocad .bti file to read and import.

Mapping file

Specify the mapping file to use. A sample mapping BOCAD_Master_Physical_Map.xml file is delivered with Smart 3D in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

Log file

Specify a log file name.

Action

Select whether to import or preview the contents of the bocad file.

- Select **Import** to import objects from the bocad into the model. This option checks the unique identification numbers in the bocad file with the identification numbers of the model objects finding matching objects. Matching objects from the bocad file are skipped during import because they already exist in the model. Only new members in the bocad file are imported into the model.
- Select **Import as copy** to import objects from the bocad into the model. This option does not compare identification numbers in the bocad file with the identification numbers of model objects. This option allows you to import the bocad objects as new, unique objects to the model.
- Select **Import and replace** to import objects from the bocad into the model. This option checks the unique identification numbers in the bocad file with the identification numbers of the model objects finding matching objects. This option deletes the existing, matching model object and places a new model object from the bocad file. Because of this delete and replace, other objects connected to the deleted and replaced object must be reconnected using Member Autoconnect.
- Select **Preview** to identify problems with the bocad file before the actual import is attempted. Objects in the bocad file that have not been imported cannot be previewed. All cross-sections, materials, and plate stocks found in the import file are written to the log file.

View Log

Opens the log file.

SECTION 22

Exporting ACIS and IGES Data

You can export geometry data in ACIS or IGES format. The export operation operates against selected sheetbody and wirebody objects.

If you are exporting in IGES format, you can set the output data units using the **Tools > Options** command. For more information about **Tools > Options**, see Options in the *Common User's Guide*.

For more information about exporting ACIS and IGES data, see *Export ACIS (File Menu)* (on page 1236) and *Export IGES (File Menu)* (on page 1236).

Export ACIS (File Menu)

File > Export > ACIS saves the selected wireframe and surface geometry data to a file in ACIS format. This command supports the output of geometry to a file but not object properties.

Export IGES (File Menu)

File > Export > IGES saves the selected wireframe and surface geometry data to a file in IGES format. This command supports the output of geometry to a file but not object properties. You must have a license for IGES in order to use this command.

SECTION 23

Exporting Stiffeners and Seams

You can export one or more complete profile stiffener or seam systems to Microsoft Excel workbook files. All stiffener and seam definition types and all data associated with each creation type are supported. Data in the workbooks can be used to import stiffeners or seams into other Model Databases, or can be changed and imported back into your current model.

See Also

Export Seam Data (on page 1239)
Export Seams (File Menu) (on page 1239)
Export Stiffener Data (on page 1238)
Export Stiffeners (File Menu) (on page 1237)

Export Stiffeners (File Menu)

File > **Export** > **Stiffeners** saves the selected stiffener data to a Microsoft Excel workbook. All stiffener definition types (by intersection, by offset, by 2D projection, and by table) can be exported. All stiffener data associated with each definition type are supported, including names, attributes, cross-section types, wireframe coordinates, boundaries, and orientations.

The output workbook uses the stiffener import-export format.xls workbook as a template. For more information, see *Stiffener Import Workbook* (on page 1218).

Export Stiffeners Ribbon

Displays the options that you use to save profile stiffener system data to a Microsoft Excel workbook. Geometry and attribute data are exported.

Select

Specifies profile stiffener systems to be exported. Selection is based on the **Filter** option.

Export

Opens the **Export Stiffeners** dialog box, which you use to specify export file names and properties. For more information, see *Export Stiffeners Dialog Box* (on page 1238).

Reject

Clears all selected stiffeners.

Accept

Accepts all selected stiffeners.

Filter

Filters the object type available with **Select**. The **Plate** option allows selection of all profile stiffeners on a plate. The **Stiffeners** option allows selection of individual profile stiffeners.

Export stiffener data

1. Click **File > Export > Stiffeners**.
2. Define the **Filter** for profile stiffener selection.
3. Select a root plate system if the **Plate** filter is defined. Select one or more stiffeners from the same plate system if the **Profiles** filter is defined.
4. Click **Export**.
5. Define the location and name of the Excel workbook in **Save As**.
6. Define the location and name of the log file in **Log File**.
7. Click **Submit**.

NOTES

- Click **Cancel** on the **Export Progress** dialog box to stop the export. Processing completes on the current stiffener, updates the log file, and closes the dialog box.
- After export finishes, click **View** to see the log file.

Export Stiffeners Dialog Box

Sets options for exporting stiffener data to a Microsoft Excel workbook.

Coordinate System Separator

Defines a character that is inserted into workbook cells between the coordinate system name and the reference plane name. The character is used when a reference plane is part of stiffener definition, such as for the intersection plane, the sketch plane, or boundaries. The default value is @.

IMPORTANT

- Choose a character that is not part of the coordinate system or reference plane names.
- Do not use the backslash character "****". This character has special meaning to the Windows operating system.

Save As

Specifies the name of and path to the new workbook file containing exported stiffener data.

- **Browse** - Uses the **Open** dialog box to select the path.
- **View** - Opens the workbook.

Log File

Specifies the name of and path to the new log file that is created during export.

- **Browse** - Uses the **Open** dialog box to select the path.
- **View** - Opens the log file.

Submit

Starts exporting stiffener data to the specified workbook.

Cancel

Cancels the command and closes the dialog box.

Close

Closes the dialog box. Use after export finishes.

See Also

Export Stiffeners (File Menu) (on page 1237)
Export Stiffeners Dialog Box (on page 1238)

Export Seams (File Menu)

File > **Export** > **Seams** saves the selected seam data to a Microsoft Excel workbook. All seam definition types (by intersection, by offset, by 2D sketch, and by table) can be exported. All seam data associated with each definition type are supported, including names, attributes, wireframe coordinates, and boundaries.

The output workbook uses the stiffener import-export format.xls workbook as a template. For more information, see *Seam Import Workbook* (on page 1226).

Export Seams Ribbon

Displays the options that you use to save profile seam data to a Microsoft Excel workbook. Geometry and attribute data are exported.

Select

Specifies seams to be exported. Selection is based on the **Filter** option.

Export

Opens the **Export Seams** dialog box, which you use to specify export file names and properties. For more information, see *Export Seams Dialog Box* (on page 1240).

Reject

Clears all selected seams.

Accept

Accepts all selected seams.

Filter

Filters the object type available with **Select**. The **Plate** option allows selection of all seams on a plate. The **Seams** option allows selection of individual seams.

Export seam data

1. Click **File > Export > Seams**.
2. Define the **Filter** for seam selection.
3. Select a root plate system if the **Plate** filter is defined. Select one or more seams from the same plate system if the **Seams** filter is defined.
4. Click **Export**.
5. Define the location and name of the Excel workbook in **Save As**.
6. Define the location and name of the log file in **Log File**.

7. Click **Submit**.

NOTES

- Click **Cancel** on the **Export Progress** dialog box to stop the export. Processing completes on the current seam, updates the log file, and closes the dialog box.
- After export finishes, click **View** to see the log file.

Export Seams Dialog Box

Sets options for exporting seam data to a Microsoft Excel workbook.

Coordinate System Separator

Defines a character that is inserted into workbook cells between the coordinate system name and the reference plane name. The character is used when a reference plane is part of seam definition, such as for the intersection plane, the sketch plane, or boundaries. The default value is @.

IMPORTANT

- Choose a character that is not part of the coordinate system or reference plane names.
- Do not use the backslash character "****". This character has special meaning to the Windows operating system.

Save As

Specifies the name of and path to the new workbook file containing exported seam data.

- **Browse** - Uses the **Open** dialog box to select the path.
- **View** - Opens the workbook.

Log File

Specifies the name of and path to the new log file that is created during export.

- **Browse** - Uses the **Open** dialog box to select the path.
- **View** - Opens the log file.

Submit

Starts exporting seam data to the specified workbook.

Cancel

Cancels the command and closes the dialog box.

Close

Closes the dialog box. Use after export finishes.

See Also

Export Seam Data (on page 1239)
Export Seams (File Menu) (on page 1239)

SECTION 24

Export Neutral/GeniE XML (File Menu)

File > **Export** > **Neutral/GeniE Export** saves the model data to a neutral or GeniE XML format. For neutral XML, data from the following types of objects are exported:

- Plate systems
- Bracket systems
- Profiles systems, which include beam systems, profile edge reinforcement systems, and stiffener systems.
- Member systems
- Openings
- Compartments
- Loadings
- Coordinate systems

For GeniE XML, data from the following types of objects are exported:

- Plate systems
- Bracket systems
- Profiles systems, which include beam systems, profile edge reinforcement systems, and stiffener systems
- Member systems
- Openings
- Slabs
- Equipment
- Equipment foundations
- Footings
- Ladders
- Stairs
- Hand rails
- Analysis loads
- Load cases
- Load combinations

NOTES

- The deactivated plate and stiffener leaf systems are skipped during export.
- Export to neutral output requires a S3N license key installed on the License server.

- Export to GeniE output requires a S3G license key installed on the License server.

This command uses two Microsoft Excel workbooks. For neutral schema output, the command uses the ExportAttributeMapping.xls workbook. By default, this workbook is delivered to the *[Reference Data Folder]*\SharedContent\Data\TranslatorsMarine\AttributeMapFile folder. For GeniE output, the command uses the S3DGeniEExportMapping.xls workbook, which is delivered to the *[Reference Data Folder]*\SharedContent\Data\TranslatorsMarine\AttributeMapFile folder. Before you export model data to either format, you must modify the associated mapping worksheet to ensure that the required user interfaces and attributes are defined. For more information, see *Export Attribute Mapping Workbooks* (on page 1246).

Supported Versions of the GeniE Software

Supported GeniE Version: 7.2-07

Export NSchema/GeniE Dialog Box **(on page 1244)**

Export data to Neutral XML

IMPORTANT Export to Neutral XML output requires a S3N license key installed on the License server.

1. Click **File** > **Export** > **Neutral/GeniE Export**.

   The Export NSchema/GeniE dialog box opens.

2. Define the **Filter** for selection.
3. Select **Neutral** from the **Output format**.
4. Specify the type of plate systems that need to be exported, and the required file type under **External Geometry Export Options** section.

   NOTES

   - The **External Geometry Export Options** exports the selected plate systems geometry to the specified external file type. The remaining plate systems geometry is exported to the output XML file.
   - If you select **Imported** and **IGES**, the software exports all of the imported plate systems geometry to an IGES file format.
   - If you select **All** and **IGES**, the software exports all of the plate systems geometry to an IGES file format.
   - If you select **Non Planar** and **IGES**, the software exports all the non planar systems geometry to an IGES file format.
   - If you select **Imported** and **ACIS**, the software exports all of the imported plate systems geometry to an ACIS file format.
   - If you select **All** and **ACIS**, the software exports all of the plate systems geometry to an ACIS file format.
   - If you select **Non Planar** and **ACIS**, the software exports all the non planar systems geometry to an ACIS file format.

5. Specify the name and location for the neutral XML file in the **XML file** box.
6. Specify the name and location for the neutral mapping file in the **Mapping file** box.

7. Specify the name and location for the log file in the **Log file** box.
8. Click **OK**.

NOTE After export finishes, click **View XML** to see the XML file. Click **View Log** to see the log file.

Export data to GeniE XML

IMPORTANT Export to GeniE output requires a S3G license key installed on the License server.

1. Click **File > Export > Neutral/GeniE Export**.

   The Export NSchema/GeniE dialog box opens.
2. Define the **Filter** for selection.
3. Select **GeniE** from the **Output format**.
4. Specify the name and location for the GeniE XML file in the **XML file** box.
5. Specify the name and location for the GeniE mapping file in the **Mapping file** box.
6. Specify the name and location for the log file in the **Log file** box.
7. Click **OK**.

NOTE After export finishes, click **View XML** to see the XML file. Click **View Log** to see the log file.

Configure the GeniE export initialization file

The GenieExportTranslator.ini file controls the export of model data to GeniE XML format. You can modify the default values to suit your specific project needs.

1. In Windows Explorer, browse to the following location:

   [Reference Data Folder]\SharedContent\Data\TranslatorsMarine\S3DGeniEExport
2. Open the *GenieExportTranslator.ini* file and modify the options as needed.
3. Save your changes.

NOTE SharedContent is delivered in the reference data setup.

GeniE Export Initialization File Global Options

DeveloperLogging

Controls the logging mode. The following values are available:

- **0** - Generates the log file in End User mode. This is the default.
- **1** - Generates the log file in Developer mode.

GeniEACISVersion

Denotes the ACIS version that GeniE software expects. The following values are available:

- **16** - Creates embedded data in ACIS version 16 format. This is the default.

- **19** - Creates embedded data in ACIS version 19 format . You can give any ACIS version that GeniE can accept.

Limitations of GeniE Export

GeniE export has the following limitations:

1. GeniE does not import overlapped plates. Before you export to ***GeniE.xml*** file, verify that the plates are not overlapped.
2. Objects with duplicate names are not imported into GeniE. Verify that the objects names are unique before you export to the GeniE xml file.
3. GeniE export does not support surface loads.
4. GeniE does not support the concept of Slab and Layer. Slab is exported as a plate and the layer material with larger thickness is exported as plate material to GeniE.

Export NSchema/GeniE Dialog Box

Sets options for exporting model data to a neutral XML format.

General

Filter

Defines the filter used to determine which objects are exported.

Description

Specifies a text description for the export.

Author

Specifies the author assigned to the export.

Organization

Specifies the organization assigned to the export.

Output Format

Neutral Schema

Indicates that the output XML is in the neutral format. Exporting to neutral schema output requires a **S3N** license key installed on the License server.

Genie

Indicates that the output XML is in the GeniE format. Exporting to GeniE output requires a **S3G** license key installed on the License server.

External Geometry Export Options

These options specify the type of plate systems to export to the selected file type.

Plate Systems

Imported

Exports all of the imported plate systems geometry to the specified file type. The remaining plate systems geometry is exported to the output XML file.

All

Exports all of the plate systems geometry to the specified file type.

Non Planar

Exports all the non planar plate systems geometry to the specified file type. The planar plate systems geometry is exported to the output XML file.

File Type

IGES

Outputs exported plate systems to an IGES file.

ACIS

Outputs exported plate systems to an ACIS file.

NOTE Plate System File options are not applicable for the GeniE output format. When you select **Genie**, these options are unavailable. By default, all the plate systems are exported to the GeniE XML file.

Paths

XML File

Specifies the name of and path to the XML file that is created during export.

Mapping File

Specifies the name of and path to the mapping file that is used during export.

Log File

Specifies the name of and path to the log file that is created during export.

Browse

Uses the **Open** dialog box to select the path.

OK

Starts exporting to the XML file, and closes the dialog box.

Cancel

Cancels the command, and closes the dialog box.

Apply

Starts exporting to the XML file, but does not close the dialog box.

View Log

Displays the log file.

View XML

Displays the XML file.

Export Attribute Mapping Workbooks

The **ExportAttributeMapping.xls** and **S3DGeniEExportMapping.xls** workbooks are used to define mapping attributes required for model data export. By default these files are delivered to the *[Reference Data Folder]*\SharedContent\Data\TranslatorsMarine\AttributeMapFile and *[Reference Data Folder]*\SharedContent\Data\TranslatorsMarine\S3DGeniEExport folders.

ExportAttributeMapping.xls (on page 1246)

S3DGeniEExportMapping.xls (on page 1250)

See Also

Export Data to Neutral XML (on page 1242)
Export NSchema/GeniE Dialog Box (on page 1244)

ExportAttributeMapping.xls

The ExportAttributeMapping.xls workbook consists of all required types of translation mapping worksheets. Before you export Smart 3D model data, you must modify these mapping worksheets to ensure that all of the necessary criteria have been met.

Worksheets

Index

Provides an index of all the other sheets in the workbook.

NSchemaMap

Provides the main mapping sheet for neutral schema mapping. This sheet in turn refers to the other NSchema mapping sheets.

Smart3DClassMap

Maps the supported classes between Smart 3D and the neutral schema. This is the main mapping sheet for the neutral schema translation.

CompartmentMap

Maps structural compartment user attributes and interfaces between Smart 3D and the neutral schema.

SpatialLoadingMap

Maps structural spatial load user attributes and interfaces between Smart 3D and the neutral schema.

UnitLoadingMap

Maps structural unit load user attributes and interfaces between Smart 3D and the neutral schema.

ModelDetailsMap

Maps model details between Smart 3D and the neutral schema.

ModelAdditionalAttributesMap

Maps model additional model attributes between Smart 3D and the neutral schema.

AdditionalAttributesMap

Maps additional model attributes between Smart 3D and the neutral schema.

PrincipalParticulars

Maps principal particulars attributes between Smart 3D and the neutral schema.

ThirdPartyDetailsMap

Maps third party application details between Smart 3D and the neutral schema.

ThirdPartyApplicationAttrMap

Maps third party application attribute details between Smart 3D and the neutral schema.

ThirdPartyAdditionalAttrMap

Maps third party additional attribute details between Smart 3D and the neutral schema.

ShipCrossSectionMap

Maps the Smart 3D cross-section user attributes and interfaces between Smart 3D and the neutral schema.

MemberSystemMap

Maps the member system user attributes and interfaces between Smart 3D and the neutral schema.

MemberPartMap

Maps the member part user attributes and interfaces between Smart 3D and the neutral schema.

DesignMemberPartMap

Maps the design member part user attributes and interfaces between Smart 3D and the neutral schema.

PlantCrossSectionMap

Maps the Smart 3D cross-section attributes and interfaces between Smart 3D and the neutral schema.

GlobalsMap

Defines global settings for data export to the neutral schema.

RootPlateSystemMap

Maps the root plate system attributes and interfaces between Smart 3D and the neutral schema.

LeafPlateSystemMap

Maps the leaf plate system attributes and interfaces between Smart 3D and the neutral schema.

RootProfileSystemMap

Maps the root profile system attributes and interfaces between Smart 3D and the neutral schema.

LeafProfileSystemMap

Maps the leaf profile system attributes and interfaces between Smart 3D and the neutral schema.

RootBracketSystemMap

Maps the root bracket system attributes and interfaces between Smart 3D and the neutral schema.

LeafBracketSystemMap

Maps the leaf bracket system attributes and interfaces between Smart 3D and the neutral schema.

RootEdgeReinforcementSystemMap

Maps the root edge reinforcement system attributes and interfaces between Smart 3D and the neutral schema.

LeafEdgeReinforcementSystemMap

Maps the leaf edge reinforcement system attributes and interfaces between Smart 3D and the neutral schema.

RootBeamSystemMap

Maps the root beam system attributes and interfaces between Smart 3D and the neutral schema.

LeafBeamSystemMap

Maps the leaf beam system attributes and interfaces between Smart 3D and the neutral schema.

CoordinateSystemMap

Maps the coordinate system attributes and interfaces between Smart 3D and the neutral schema.

OpeningMap

Maps the opening attributes and interfaces between Smart 3D and the neutral schema.

Mapping Sheets Columns

Source Attribute

Defines the Smart 3D object attribute to map.

Target Attribute

Defines the neutral schema attribute to which the Smart 3D attribute value is mapped.

MapType

Defines the map type of the attribute. Acceptable values are as follows:

- **NameDef** - Used when the value needs to be directly transferred. For example, there is an attribute named *Name* in the source system and the same attribute named *name* in the target system. Because both attributes have the same meaning, the value just needs to be transferred between the two systems. By default, if no **MapType** is specified, **NameDef** is used.
- **ValueDef** - Used when the value of an attribute needs to be mapped before it is transferred.
- **AdditionalAttributeDef** - Used when it is necessary to create new or additional attributes for the value of a given attribute.

- **PatternDef** - Used when any valid regular expression can be used to map the values.
- **CharacterDef** - Used to split the value of an attribute using a delimiter or character position and string length to generate new attributes.

SheetName

Defines the name of the sheet in the ExportAttributeMapping.xls workbook that contains additional mapping information related to the attribute, if any exists.

Formula

Defines the formula used to calculate the value of the attribute to mapped, if one is needed. For example, if the source system has an attribute Radius that needs to be mapped to the destination system's attribute Diameter, the software cannot substitute one value for the other. Instead the value used for Diameter results from the following calculation: Diameter = 2 * Radius.

Value

Specifies the value of the neutral schema attribute.

Label

Specifies whether the Smart 3D attribute is a label. Type **TRUE/YES** or **FALSE/NO**. Entries can be in lowercase or uppercase.

Ignore

Specifies whether the item is ignored during export. Type **TRUE/YES** or **FALSE/NO**. Entries can be in lowercase or uppercase.

Mapping Tips

Use the labels in Smart 3D to map the direct properties of objects or the properties of the related objects such as traverse relationships.

The following steps illustrate how to export the structure member weight to the neutral schema xml file:

1. If a label is not available in the Catalog task, create a label that gives the **Structure Member Weight** of the placed member.

   NOTE For more information about creating labels, see *Label Editor Command* in the *Catalog User's Guide*.

2. Open the **MemberPartMap** sheet in translation mapping file.

   NOTE The **MemberPartMap** sheet is available in the **ExportAttributeMapping.xls** workbook. By default, the **ExportAttributeMapping.xls** workbook is available in the following location:

   [Reference Data Folder]\SharedContent\Data\TranslatorsMarine\AttributeMapFile.

3. Map the Smart 3D Attribute to the corresponding Neutral Schema Attribute.
4. Type **TRUE** in the **Label** column.
5. Save the mapping file.

NOTE Label mapping is not applicable for GeniE export functionality.

Hierarchical Mapping

The **ExportAttributeMapping.xls** workbook also supports hierarchical mapping between source and target systems. For example, if the source object has an attribute value called *web length* on a cross-section object using the following hierarchy:

IJUAXSectionWeb.WebLength

If the above hierarchy is matched with *Web.WebLength* on the target system, then you can directly map this hierarchy in the **Value** and **Target Attribute** columns.

NOTE This hierarchical mapping is applicable until a child item is a direct attribute on its parent.

S3DGeniEExportMapping.xls

This workbook contains mapping for both neutral schema and GeniE export. That is, it contains all the sheets of the **ExportAttributeMapping.xls** workbook and the mapping sheets needed for GeniE export. In the existing sample mapping sheets, you can add new mapping entries.

Worksheets

Index

Provides an index of all the other sheets in the workbook. It describes all other sheets in the workbook.

GeniEMap

Provides the main mapping sheet for GeniE mapping. This sheet in turn refers to the other GeniE mapping sheets.

GeniEClassMap

Maps the supported structure classes between Smart 3D and GeniE export. This is the main mapping sheet for the GeniE translation.

GeniEGlobalsMap

Defines global settings for data export to GeniE Schema.

StraightMemSystemAttrsMap

Maps the straight member system attributes and interfaces between the neutral and GeniE schemas.

CurvedMemSystemAttrsMap

Maps the curved member system attributes and interfaces between the neutral and GeniE schemas.

StraightMemPartAttrsMap

Maps the straight member part attributes and interfaces between the neutral and GeniE schemas.

CurvedMemPartAttrsMap

Maps the curved member part attributes and interfaces between the neutral and GeniE schemas.

StraightStiffenerSysAttrsMap

Maps the straight stiffener system attributes and interfaces between the neutral and GeniE schemas.

CurvedStiffenerSysAttrsMap

Maps the curved stiffener system attributes and interfaces between the neutral and GeniE schemas.

StraightERSysAttrsMap

Maps the straight edge reinforcement system attributes and interfaces between the neutral and GeniE schemas.

CurvedERSysAttrsMap

Maps the curved edge reinforcement system attributes and interfaces between the neutral and GeniE schemas.

BeamSysAttrsMap

Maps the beam system attributes and interfaces between the neutral and GeniE schemas.

MaterialMap

Maps the material attributes and interfaces between the neutral and GeniE schemas.

ApplicationDetailsMap

Maps the application details between the neutral and the GeniE schemas.

UnitsNameMap

Maps the unit names and interfaces between the neutral and GeniE schemas.

UnitsMap

Maps the unit attributes and interfaces between the neutral and GeniE schemas.

FlatBarXSectionMap

Maps the flat bar cross-section attributes and interfaces between the neutral and GeniE schemas.

AngleXSectionMap

Maps the angle cross-section attributes and interfaces between the neutral and GeniE schemas.

GeneralXSectionMap

Maps the generic cross-section attributes and interfaces between Neutral and GeniE Schema.

IXSectionMap

Maps the I-cross-section attributes and interfaces between the neutral and GeniE schemas.

ChannelXSectionMap

Maps the channel cross-section attributes and interfaces between the neutral and GeniE schemas.

TubeXSectionMap

Maps the tubular cross- section attributes and interfaces between the neutral and GeniE

schemas.

BoxXSectionMap

Maps the box cross-section attributes and interfaces between the neutral and GeniE schemas.

XSectionAdditionalAttrMap

Maps the cross-sectional additional attributes and interfaces between the neutral and GeniE schemas.

SlabAttrsMap

Maps the slab attributes and interfaces between the neutral and GeniE schemas.

FlatPlateSysAttrsMap

Maps the flat plate system attributes and interfaces between the neutral and GeniE schemas.

CurvedPlateSysAttrsMap

Maps the curved plate system attributes and interfaces between the neutral and GeniE schemas.

EquipmentAttrsMap

Maps the equipment attributes and interfaces between the neutral and GeniE schemas.

EqpFoundationAttrsMap

Maps the equipment foundation attributes and interfaces between the neutral and GeniE schemas.

FootingAttrsMap

Maps the footing attributes and interfaces between the neutral and GeniE schemas.

StairAttrsMap

Maps the stair attributes and interfaces between the neutral and GeniE schemas.

LadderAttrsMap

Maps the ladder attributes and interfaces between the neutral and GeniE schemas.

HandRailAttrsMap

Maps the hand rail attributes and interfaces between the neutral and GeniE schemas.

LoadCaseAttrMap

Maps the load case attributes and interfaces between the neutral and GeniE schemas.

LoadCombinationAttrMap

Maps the load combination attributes and interfaces between the neutral and GeniE schemas.

PointLoadAttrMap

Maps the point load attributes and interfaces between the neutral and GeniE schemas.

UDLAttrMap

Maps the UDL load attributes and interfaces between the neutral and GeniE schemas.

Mapping Sheets Column Description

Source Attribute

Defines the neutral schema attribute to be mapped.

Target Attribute

Defines the GeniE schema attribute to which the neutral schema attribute value is mapped.

MapType

Defines the map type of the attribute. Acceptable values are as follows:

- **NameDef** - Used when the value needs to be directly transferred. For example, there is an attribute named *Name* in the source system and same attribute named *name* in the target system. Because both attributes have the same meaning, the value just needs to be transferred between the two systems. By default, if no **MapType** is specified, **NameDef** is used.
- **ValueDef** - Used when the value of an attribute needs to be mapped before it is transferred.
- **AdditionalAttributeDef** - Used when it is necessary to create new or additional attributes for the value of a given attribute.
- **PatternDef** – Used when any valid regular expression can be used to map the values.
- **CharacterDef** - Used to split the value of an attribute using a delimiter or character position and string length to generate new attributes.

SheetName

Defines the name of the sheet in the ExportAttributeMapping.xls workbook that contains additional mapping information related to the attribute, if any exists.

Formula

Defines the formula used to calculate the value of the attribute to mapped, if one is needed. For example, if the source system has an attribute Radius that needs to be mapped to the destination system's attribute Diameter, the software cannot substitute one value for the other. Instead the value used for Diameter would result from the following calculation: Diameter = 2 * Radius.

Value

Specifies the value of the neutral schema attribute.

Label

Specifies whether the Smart 3D attribute is a label. Type **TRUE/YES** or **FALSE/NO**. Entries can be in lowercase or uppercase.

Ignore

Specifies whether the item is ignored during export. Type **TRUE/YES** or **FALSE/NO**. Entries can be in lowercase or uppercase.

Hierarchical Mapping

The **S3DGeniEExportMapping.xls** workbook also supports hierarchical mapping between source and target systems. For example, if the source object has an attribute called *COG* on Equipment using the following hierarchy:

EquipmentProperties.Physical.DryCoG.Point3D.X

And if the above hierarchy is matched with *cog.x* on the target system, then you can directly map this hierarchy in the **Source Attribute** and **Target Attribute** columns.

NOTE This hierarchical mapping is applicable until a child item is a direct attribute on its parent.

SECTION 25

Copy by Family

The **Edit > Copy by Family** command replicates structural objects on the **Select Copy Family** dialog box using three methods:

- **Copy Similar** replicates major portions of the model with the ability to automatically update the replicas as changes are made. A copy similar family consists of groups of structure with similar grid planes. One group is copied to the other groups. A typical example is to define a transverse bulkhead, including profile stiffeners, seams, and openings, and copy the bulkhead to other transverse locations.
- **Copy Symmetry** replicates portions of the model across a longitudinal centerline symmetry grid plane. A copy symmetry family consists of groups of structure on both the port and starboard sides of the symmetry plane. One group is reflected about the symmetry plane to the other group.
- **Copy Across Models** replicates major portions of a model to another model.

NOTE Within the **Create Target Group** and **Modify Target Group**, you can manually select objects to copy. For more information, see *Manually copy objects* (on page 1271).

Copy by Family Batch Services

Smart 3D uses the SmartPlant Batch Services application to help you perform **Copy by Family** functions in the background. You can submit the batch jobs that are executed in a batch queue. For this, you must create, enable, and start batch queues before submitting the batch jobs.

NOTE For more information about SmartPlant Batch Services, see the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

You can use the following guidelines listed to run the batch services:

- SmartPlant Batch Services must be installed on the client workstation that submits a batch job and on the batch server workstation that processes the job. The necessary software queues must also be created. For information on installing SmartPlant Batch Services and setting up the batch queues, see *Installing SmartPlant Batch Services* in the *Smart 3D Installation Guide*.
- The batch jobs that are submitted using this functionality can only be monitored by the SmartPlant Batch Services application.
- Use the **Intergraph Batch Manager** tool provided by the Intergraph Batch Services to discontinue any batch job. The **Batch Timeout** property specified on the **Property** page is no longer valid.
- The mapped account that performs the batch job does not need to be an administrator on the batch server.
- The batch processes run as a Windows service using a configured login. You do not need to be logged on to the batch server.
- In the **Intergraph Batch Server Properties (Local Computer)** > **Log On**, make sure **Allow service to interact with desktop** is selected.

- For Windows XP batch servers, grant Write permissions for each mapped account to the %SYSTEMDRIVE%/Documents and Settings/Default User/Application Data/Microsoft/AddIns folders.
- For 64-bit batch servers, the %windir%/SysWOW64/config/systemprofile/desktop folder must be manually created and every mapped account must have Write access on that folder. For most machines, the %windir% folder is be located in the C:/Windows. However this may vary depending on your Windows installation.
- For batch jobs, Microsoft Excel must have the same security settings that are necessary for interactive updates. Configure Microsoft Excel for every SmartPlant Batch Services mapped account on the batch server, so that it can be accessed by the batch processes. For more information, see *Reports User's Guide.*
- Microsoft Excel must be opened on the batch server at least once by each SmartPlant Batch Services mapped user. Also, **Trust Settings** for Microsoft Excel must be included.
- For Windows 7 batch servers, the registry key HKEY_USERS/.DEFAULT/Software/Microsoft/VBA must be manually created with Write permissions given individually for each mapped account.

Select Copy Family Dialog Box

Toolbar


Create New Similar Family

Creates a new copy similar family. For more information on creating copy similar families, see *Similar Family Properties Dialog Box* (on page 1276).


Create New Symmetry Family

Creates a new symmetry family. For more information on creating copy symmetry families, see *Symmetry Family Properties Dialog Box* (on page 1279).


Create New Across Model Family

Creates a new across model family. For more information on creating across model families, see *Across Model Family Definition Dialog Box* (on page 1281).


Delete Family Definition

Deletes the selected family.


Rename Family Definition

Renames the selected family. You must select a family before you can rename that family.


Edit Family Definition

Edits the selected family. You must select a family before you can edit that family.


Refresh Family

Removes any relationships created in the family by using the **Synchronize** command and the **Copy by Family** ribbon. For example, **Refresh Family** restores highlighting of objects in the graphics view for the **Find and correlate objects** step if that step has been previously carried out.

Copy Shortcuts

Configures copy shortcut keys for similar or symmetry families and specify a log file to store the copy operation. For more information, see *Copy Shortcuts Settings Dialog Box* (on page 1284).

Copy Family Hierarchy

Copy Across Model Families

Lists the Copy Across Model Families that you have defined.

Copy Similar Families

Lists the copy similar families that you have defined.

Copy Symmetry Families

Lists the copy symmetry families that you have defined.

Batch

Displays the **Copy by Family Batch** dialog box. You can use this dialog box to execute jobs in a batch queue. For more information, see *Copy By Family Batch Dialog Box* (on page 1285).

Synchronize

Opens the **Copy by Family** ribbon for the selected family.

Close

Closes the dialog box.

Copy by Family Ribbon

The **Copy by Family** ribbon copies structural objects between the source and target family groups, and keeps the family groups in synchronization. Similar families and symmetry families both use the ribbon. The ribbon is available after clicking **Synchronize** on the **Select Copy Family** dialog box.

Find and correlate objects

Select to highlight similar or symmetrical structural objects that meet the criteria of both the **Source Group** and **Target Group** of the Similar or Symmetry family. When you click **Finish**, the software adds the objects to the copy family so that duplicates are not created during the copy. Select **Find Man** to manually add objects that were not found automatically.

The search for similar/symmetry objects first considers if they have the same inputs (that is, the same boundaries, supporting frame, and so on). It then matches geometry (for example, starboard plate is the same as the mirrored port plate). For similar relationships, parametric changes caused by the hull are recognized.

Because this is a strong matching criterion, multiple finds in the same group are typically caused by the same object being placed twice. These overlapping finds are reported as errors for you to fix, and the similar/symmetry relation is not made.

Because this is such a strong criteria, the **Manual Find** option allows you to force objects to be similar that are not similar according to the automatic criteria. After using **Manual Find**, you can run **Find and correlate objects** to match objects that use the manually similar entities as inputs.

Use the **Find and correlate objects** option before a copy or modify operation if there is a chance that a similar/symmetry object has been created in the target area after initially using the **Copy by Family** command.

Potential similar or symmetry objects can be created in the target area by the following operations:

- Creating a new object in the target area.
- The copy result of a different family copies an object into the same target area.
- An automatic process creates a new object in the target area. For example, the execute split command creates intersection seams in the target area that are similar to the source area.

 **Create target group copy**

Select to highlight objects in the source group that are not in the target group. The software automatically finds all plate systems that use the grid plane of the source group as the surface definition. The software then finds all children of the plate system (seams, stiffeners, openings, and features) and all the structural objects that are bounded by either the grid plane or the plate systems defined by the grid plane and all their children. You can click objects in the graphics view to clear them and prevent them from being copied from the source group to the target group. Select **Browser** to clear objects in a hierarchical list. The software displays the cleared objects in green in both the graphics view and the Browser. When you click **Finish**, all highlighted objects are copied to create the target group.

 **Modify target group copy**

Select to highlight objects that are different between the source and target groups. If a family for which a copy has already been created contains changed copied objects or new objects, use this option to update the target group copy. You can click objects in the graphics view to clear them and prevent them from being updated. Select **Browser** to clear objects in a hierarchical list. The software displays the cleared objects in green in both the graphics view and the Browser. When you click **Finish**, all highlighted objects are updated in the target group.

 **Delete target group copy**

Select to highlight objects that have been previously copied and deleted from the source group, but remain in the target group. You can click objects in the graphics view to clear them and prevent them from being deleted in the target group. The software marks these cleared objects as recognized differences between the groups and displays the objects in green. Select **Browser** to clear objects in a hierarchical list. The software displays the cleared objects in green in both the graphics view and the Browser. When you click **Finish**, all highlighted objects are deleted from the target group.

Finish

Performs the copy create, modify, or delete as appropriate.

 **Reject**

Resets the selected objects.

 **Accept**

Confirms that the selected objects are the ones you want to work on.

Family

Select the family to work with. You can create new families by using the **Create New Similar Family** or **Create New Symmetry Family** commands on the **Select Copy Family** dialog box. For more information, see *Select Copy Family Dialog Box* (on page 1283).

Source Filter

Select the filter to be used by the create or modify operations.

TIPS

- When you copy or update objects in a similar or symmetry family, the **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the Copy by Family operation is performed on only those objects satisfying both the predefined filter and the Workspace filter. When the **<Filter> + Workspace Filter** option is in use, the **Source Filter** box is highlighted in yellow, and the **Find** and **Delete** buttons are unavailable. The **<Filter> + Workspace Filter** option supports only similar and symmetry families.
- When you copy across models, you can select the **Select Filter** option to display all filters in the model corresponding to those in the source model of the selected copy across model family. Select the filter you need from the **Select Filter** dialog box. Your filter selection is remembered in the session file and is used as the default source filter the next time you copy across models. For more information, see *Select Filter Dialog Box* (on page 1282).

Source Group

Select the family group to use as the copy source.

Target Group

Select one or more family groups to create or modify.

Browser

Opens a browser similar to the **Workspace Explorer** that lists objects included in the target group copy. You can clear objects from the browser. The **CDF** tab (Copy Dependent Full) lists the objects in a hierarchy. The **CDT** tab (Copy Dependent by Type) sorts the objects by type. You can de-select objects from the browser. The software displays selected objects in yellow and cleared objects in green.

Family Synchronize Options

Defines find, create, and modify options for synchronizing the family. For more information, see *Synchronize Options Dialog Box* (on page 1275).

What do you want to do?

- *Create a copy similar family using a single grid plane* (on page 1260)
- *Create a copy similar family using multiple grid planes* (on page 1261)
- *Modify a copy similar family* (on page 1262)
- *Delete a copy similar family* (on page 1262)
- *Create a copy symmetry family* (on page 1262)
- *Modify a copy symmetry family* (on page 1263)

- *Delete a copy symmetry family* (on page 1263)
- *Create a copy across models family using a single grid plane* (on page 1263)
- *Create a copy across models family using multiple grid planes* (on page 1264)
- *Create a copy across models family by range* (on page 1264)
- *Modify a copy across model family* (on page 1265)
- *Delete a copy across model family* (on page 1265)
- *Copy objects in a similar family* (on page 1265)
- *Update objects in a similar family* (on page 1266)
- *Delete objects in a similar family* (on page 1267)
- *Copy objects in a symmetry family* (on page 1269)
- *Update objects in a symmetry family* (on page 1269)
- *Delete objects in a symmetry family* (on page 1270)
- *Manually copy objects* (on page 1271)
- *Copy objects using Copy Shortcuts* (on page 1271)
- *Copy objects in an across model family* (on page 1272)
- *Find and correlate objects across models* (on page 1273)
- *Select the source filter for copy across models* (on page 1273)
- *Schedule a Copy by Family operation as a batch job* (on page 1274)
- *Configure the Smart 3D batch queues* (on page 1274)
- *Configure account mapping for batch services* (on page 1275)

Create a copy similar family using a single grid plane

1. Click **Edit > Copy by Family**.
2. Click **Create New Similar Family**  in the **Select Copy Family** dialog box.
   *The **Similar Family Properties** dialog box displays.*
3. Type a unique name for the family in the **Name** box.
4. Select the definition method for the family: **Standard**, **By Range**, or **Overlapping**.
5. Select the box in the grid under the **.G 1** group heading.
6. In the **Workspace Explorer**, select the grid plane that defines the objects in **.G 1**.
7. Select the box in the grid under the **.G 2** group heading.
8. In the **Workspace Explorer**, select the grid plane that defines the location of **.G 2**.
9. Optionally, click **Insert group** if you need more than two groups in the family.
10. Define the grid planes for any new groups as you did for **.G 2**.

11. Click **OK**.

NOTES

- By default, **.G 1** is the copy source group for the family (as indicated by the check in the group heading). You can specify another group to be the copy source by selecting the checkbox in that group's heading.
- All structural objects that are related to the plane in a group are included in the group after **Synchronize** is performed. For more information, see *Select Copy Family Dialog Box* (on page 1283).
- You can select a group column heading box (for example, **.G 1**) and rename the group.

Create a copy similar family using multiple grid planes

1. Click **Edit > Copy by Family**.
2. Click **Create New Similar Family**  in the **Select Copy Family** dialog box.

   *The **Similar Family Properties** dialog box displays.*
3. Type a unique name for the family in the **Name** box.
4. Select the definition method for the family: **Standard**, **By Range**, or **Overlapping**.
5. Select the box in the grid under the **.G1** group heading.
6. In the **Workspace Explorer**, select the first grid plane that defines the objects in **.G1**.
7. Select the box in the grid under the **.G2** group heading.
8. In the **Workspace Explorer**, select the grid plane that defines the location of **.G2**.
9. Click **Insert corresponding item (row)**.
10. Select the new (second) box in the grid under the **.G1** group heading.
11. In the **Workspace Explorer**, select the second grid plane that defines the objects in **.G1**.
12. Select the new (second) box in the grid under the **.G2** group heading.
13. In the **Workspace Explorer**, select the second grid plane that defines the location of **.G2**.
14. Optionally, click **Insert group** if you need more than two groups in the family.
15. Define the grid planes for any new groups as you did for **.G2**.
16. Click **OK**.

NOTES

- By default, **.G1** is the copy source group for the family (as indicated by the check in the group heading). You can specify another group to be the copy source by selecting the checkbox in that group's heading.
- All structural objects that are related to both planes in a group are included in the group after **Synchronize** is performed. For more information, see *Select Copy Family Dialog Box* (on page 1283).
- You can select a group column heading box (for example, **.G1**) and rename the group.

Modify a copy similar family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to modify under the **Copy Similar Families** hierarchy.
3. Click **Edit Family Definition** in the **Select Copy Family** dialog box. This activates the **Similar Family Properties** dialog box.
4. Using the dialog box controls, edit the family as needed.

IMPORTANT

- The grid planes in a row cannot be modified. You can delete the group with grid planes to change, and add a new group.
- An existing row cannot be deleted. You can delete the family in the **Select Copy Family** dialog box and create a new one with the needed number of rows.

Delete a copy similar family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to delete under the **Copy Similar Families** hierarchy.
3. Click **Delete Family Definition**.

Create a copy symmetry family

1. Click **Edit > Copy by Family**.
2. Click **Create New Symmetry Family** in the **Select Copy Family** dialog box.

   *The **Symmetry Family Properties** dialog box.*

3. Type a unique name for the family in the **Name** box.
4. In the **Workspace Explorer**, select one or more coordinate systems to define the symmetry plane.

   *The coordinate systems display in the **Symmetry Planes** box.*

   All selected coordinate systems must have coincident Y=0 planes.

5. In the **Port Group Name** box, type a name for the port side group.
6. In the **Starboard Group Name** box, type a name for the starboard side group.
7. Under **Source Group**, select whether the port side or the starboard side is the copy source group.
8. Click **OK**.

Modify a copy symmetry family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to modify under the **Copy Symmetry Families** hierarchy.
3. Click **Edit Family Definition** in the **Select Copy Family** dialog box.

   *The **Symmetry Family Properties** dialog box displays.*

4. Using the dialog box controls, edit the family as needed.

IMPORTANT An existing coordinate system in the **Symmetry Planes** box cannot be deleted. You can delete the family in the **Select Copy Family** dialog box and create a new one with the needed coordinate system.

Delete a copy symmetry family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to delete under the **Copy Symmetry Families** hierarchy.
3. Click **Delete Family Definition** .

Create a copy across models family using a single grid plane

1. Click **Edit** > **Copy by Family**.
2. Click **Create New Across Model Family** in the **Select Copy Family dialog** box.

   *The **Across Model Family Definition** dialog box displays.*

3. Type a name for the family in the **Name** box, or use the default name.
4. Type a value in the **Range Growth Value** box, or use the default value. The **Method** definition is selected as **Standard** by default.
5. To change the **Group Name**, click the cell corresponding to the target group name, and type the name.
6. To select the source ship, click the cell **Ship Name** of the source, and select the ship from the list. To change the **Group Name**, click the cell corresponding to the source group name, and type the name.
7. Click the cell **Corresponding Elements1** under the **Target** or **Source,** and select the grid plane that defines the target or source group.
8. Click **OK**.

NOTE All structural objects that are related to the plane in a group are included in the group after **Synchronize** is performed. For more information, see *Select Copy Family Dialog Box* (on page 1283).

Create a copy across models family using multiple grid planes

1. Click **Edit** > **Copy by Family**.
2. Click **Create New Across Model Family** in the **Select Copy Family dialog** box.

   *The **Across Model Family Definition** dialog box displays.*
3. Type a name for the family in the **Name** box, or use the default name.
4. Type a value in the **Range Growth Value** box, or use the available value. The **Method** definition is selected as **Standard** by default.
5. To change the **Group Name**, click the cell corresponding to the target group name, and type the name.
6. To select the source ship, click the cell **Ship Name** of the source, and select the ship from the list. To change the **Group Name**, click the cell corresponding to the source Group Name, and type the name.
7. Click the cell **Corresponding Elements1** under the **Target** or **Source,** and define the grid plane that defines the target/source group.
8. Click the **Add Corresponding Elements** button to add **Corresponding Elements** rows to define additional grid planes for the target or source group.
9. Click **OK**.

Create a copy across models family by range

1. Click **Edit** > **Copy by Family**.
2. Click **Create New Across Model Family** in the **Select Copy Family** dialog box.

   *The **Across Model Family Definition** dialog box displays.*
3. Type a name for the family in the **Name** box, or use the default name.
4. Type a value in **Range Growth Value** box, or use the default value.
5. Select **Method** from the **By Range** list.
6. To change the **Group Name**, click the cell corresponding to the target group name, and type the name.
7. To select the source ship, click the cell **Ship Name** of the source, and select the ship from the list. To change the **Group Name**, click the cell corresponding to the source group name, and type the name.
8. Click the cells **Corresponding Elements1** and **Corresponding Elements2** under the **Target** or **Source,** and define the grid planes for the target or source group.

9. Click **Add Corresponding Elements** to add **Corresponding Elements** rows to define additional grid planes for the target/source group. Because there are only two corresponding elements allowed when the **Method** definition is **By Range**, the **Add corresponding Elements** and **Delete Corresponding Elements** buttons are not available.
10. Click **OK**.

Modify a copy across model family

1. Click **Edit** > **Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to modify under the **Copy Across Model Families** hierarchy.
3. Click **Edit Family Definition** in the **Select Copy Family** dialog box.

   *The **Copy Across Model Family Properties** dialog box displays.*

4. Using the dialog box controls, edit the family definition as needed.

   TIPS

   - You can edit the name of the family and the range growth value.
   - Optionally, you can add additional **Corresponding Elements** rows to define additional grid planes using **Add Corresponding Elements**.

5. Click **Apply** or **OK** to save the changes.

IMPORTANT

- You cannot change the **Method** definition.
- The target and source group details cannot be edited.
- For more information on configuration details, see *Across Model Family Definition Dialog Box* (on page 1281).

Delete a copy across model family

1. Click **Edit** > **Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to delete under the **Copy Across Model Families** hierarchy.
3. Click **Delete Family Definition**.

Copy objects in a similar family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to work with under the **Copy Similar Families** hierarchy.

TIP For instructions on creating a new copy similar family, see *Create a copy similar family using a single grid plane* (on page 1260) and *Create a copy similar family using multiple grid planes* (on page 1261).

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the filter that the copy uses to find structural objects. By default, **Workspace Filter** is used.

   TIP The **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the operation is performed on only those objects satisfying both the predefined filter and the Workspace filter.

5. In the **Source Group** box, select the group from which to copy objects. By default, this is the group indicated as the source group when you defined the family. For more information, see *Similar Family Properties Dialog Box* (on page 1276).

6. In the **Target Group** box, select the group to which to copy objects from the **Source Group**. Optionally, select **Multiple** to copy to more than one target group.

7. Click **Find and correlate objects** .

   *The software highlights all existing structural objects that are similar and meet the criteria of both the **Source Group** and the **Target Group**.*

8. Click **Create target group copy** on the ribbon.

   *The software highlights in yellow all the objects in the **Source Group** that are not in the **Target Group**.*

9. Optionally, select any highlighted objects in the graphics view that you do not want to copy to the **Target Group**.

   The color of the highlighted objects changes from yellow to cyan.

10. Click **Finish**.

    *The software copies the highlighted objects in the **Source** group to the **Target** group.*

Update objects in a similar family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to update under the **Copy Similar Families** hierarchy.

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the filter that the copy uses to find structural objects. By default, **Workspace Filter** is used.

   TIP The **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the operation is performed on only those objects satisfying both the predefined filter and the Workspace filter.

5. In the **Source Group** box, select the group from which to copy objects. By default, this is the group indicated as the source group during the original copy. For more information, see *Copy Objects in a Similar Family* (on page 1265).
6. In the **Target Group** box, select the group to which to copy objects from the **Source Group**. Optionally, select **Multiple** to copy to more than one target group.
7. Click **Find and correlate objects** .

   *The software highlights all existing structural objects that are similar, meet the criteria of both the **Source Group** and the **Target Group**, and have been added after the original copy.*
8. Click **Modify target group copy** on the ribbon.

   *The software highlights all the objects that are in the **Source Group** but not in the **Target Group**, in both the **Source Group** and the **Target Group** but have changed, or have been added after the original copy.*
9. Optionally, select any highlighted objects in the graphics view that you do not want to copy to the **Target Group**.

   The color of the highlighted objects changes from yellow to green.
10. Click **Finish**.

    *The software copies the highlighted objects in the **Source** group to the **Target** group.*

NOTE Within the **Create Target Group** and **Modify Target Group**, you can manually select objects to copy. See *Manually Copy Objects* (on page 1271).

Delete objects in a similar family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*
2. Select the family to work with under the **Copy Similar Families** hierarchy.
3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*
4. In the **Source Group** box, select the group in which objects have been deleted. By default, this is the group indicated as the source group during the original copy. For more information, see *Copy objects in a similar family* (on page 1265).
5. In the **Target Group** box, select the group in which to remove the deleted objects from the **Source Group**. Optionally, select **Multiple** to copy to more than one target group.
6. Click **Delete target group copy** on the ribbon.

   *The software highlights all the objects in the **Target Group** that have been deleted from the **Source Group**.*
7. Optionally, select any highlighted objects in the **Target** group that you do not want deleted.

   The color of the highlighted objects changes from yellow to green.
8. Click **Finish**.

   *The software deletes the highlighted objects in the **Target** group. Objects highlighted in green are* not *deleted from the **Target Group**. The software marks the green objects as*

recognized differences between the two groups and excludes the objects from future copy operations.

Copy objects in a symmetry family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to work with under the **Copy Symmetry Families** hierarchy.

   TIP For instructions on creating a new copy symmetry family, see *Create a copy symmetry family* (on page 1262).

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the filter that the copy uses to find structural objects. By default, **Workspace Filter** is used.

   TIP The **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the operation is performed on only those objects satisfying both the predefined filter and the Workspace filter.

5. In the **Source Group** box, select the group from which to copy objects. By default, this is the group indicated as the source group when you defined the family. For more information, see *Symmetry Family Properties Dialog Box* (on page 1279).

6. In the **Target Group** box, select the group to which to copy objects from the **Source Group**.

7. Click **Find and correlate objects**.

   The software highlights all existing structural objects that have symmetry.

8. Click **Create target group copy** on the ribbon.

   *The software highlights in yellow all the objects in the **Source Group** that are not in the **Target Group**.*

9. Optionally, select any highlighted objects in the graphics view that you do not want to copy to the **Target Group**.

   The color of the highlighted objects changes from yellow to cyan.

10. Click **Finish**.

    *The software copies the highlighted objects in the **Source** group to the **Target** group.*

Update objects in a symmetry family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to update under the **Copy Symmetry Families** hierarchy.

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the filter that the copy uses to find structural objects. By default, **Workspace Filter** is used.

TIP The **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the operation is performed on only those objects satisfying both the predefined filter and the Workspace filter.

5. In the **Source Group** box, select the group from which to copy objects. By default, this is the group indicated as the source group during the original copy. For more information, see *Symmetry Family Properties Dialog Box* (on page 1279).
6. In the **Target Group** box, select the group to which to copy objects from the **Source Group**.
7. Click **Find and correlate objects** .

   *The software highlights all existing structural objects that are similar, meet the criteria of both the **Source Group** and the **Target Group**, and have been added after the original copy.*
8. Click **Modify target group copy** on the ribbon.

   *The software highlights all the objects that are in the **Source Group** but not in the **Target Group**, in both the **Source Group** and the **Target Group** but have changed, or have been added after the original copy.*
9. Optionally, select any highlighted objects in the graphics view that you do not want to copy to the **Target Group**.

   The color of the highlighted objects changes from yellow to green.
10. Click **Finish**.

    *The software copies the highlighted objects in the **Source** group to the **Target** group.*

NOTE Within the **Create Target Group** and **Modify Target Group**, you can manually select objects to copy. See *Manually copy objects* (on page 1271).

Delete objects in a symmetry family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*
2. Select the family to work with under the **Copy Symmetry Families** hierarchy.
3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*
4. In the **Source Group** box, select the group in which objects have been deleted. By default, this is the group indicated as the source group during the original copy. For more information, see *Symmetry Family Properties Dialog Box* (on page 1279).
5. In the **Target Group** box, select the group in which to remove the deleted objects from the **Source Group**.
6. Click **Delete target group copy** on the ribbon.

   *The software highlights all the objects in the **Target Group** that have been deleted from the **Source Group**.*
7. Optionally, select any highlighted objects in the **Target** group that you do not want deleted.

   The color of the highlighted objects changes from yellow to green.
8. Click **Finish**.

*The software deletes the highlighted objects in the **Target** group. Objects highlighted in green are* not *deleted from the **Target Group**. The software marks the green objects as recognized differences between the two groups and excludes the objects from future copy operations.*

Manually copy objects

When using the **Create target group copy** and **Modify target group copy** commands on the **Copy by Family** ribbon, you can manually select items to copy. Select from the following two methods:

Copy Preselected Objects

1. Pre-select the needed objects.
2. Click **Edit** > **Copy by Family**.
3. To create a target, select **Create target group copy**, or click **Finish**.

   Preselected objects are copied. Objects on which the selected objects are dependent are also copied.
4. If you want to modify an existing target, select **Modify target group copy**, or click **Finish**.

   Preselected objects are copied. Objects on which the selected objects are dependent are also copied.

NOTE Only the target group is modified, even if you select target and source objects.

Clear Objects and Select Again

1. Click **Edit** > **Copy by Family.**
2. Clear the **Select All** checkbox, and select **Create target group copy**.

   All the copy candidates are highlighted in green, indicating that they are no longer selected.
3. Manually select the objects to copy.

Copy objects using Copy Shortcuts

You can replicate objects under similar or symmetry families and update the replicas using customized shortcut keys.

Configure Copy Shortcuts

1. Define the shortcut keys in the CopybyFamilyShortCutKeys.txt file. The file is located in the [*Product Folder]*\CopybyFamily\Data folder.
2. Click **Edit** > **Copy by Family**.
3. In the **Select Copy Family** dialog box, click **Copy Shortcuts**.

   *The **Copy Shortcuts Settings** dialog box displays.*
4. Configure each shortcut key as follows:

a. In the **Family** cell, specify **Similar** or **Symmetry**.

b. In the **Source Group** cell, specify the family group to use as the copy source.

c. In the **Target Group** cell, specify the family group to create or modify.

d. In the **Select Child** cell, specify **True** to include child objects in the copy operation, or **False** to exclude them. The default value comes from the **Copy by Family** ribbon.

e. In the **Molded Form Only** cell, specify **True** if the shortcut key is used only in the **Molded Forms** task, or **False** if it is used across tasks. The default value comes from the **Copy by Family** ribbon.

5. In **Log File Path**, specify a log file for storing the copy shortcuts operation.
6. Click **OK**.

Replicate Objects Using Shortcut Keys

1. In the **Workspace Explorer**, select the objects under the family range to be replicated or to update their replicas.
2. Press the assigned copy shortcut key to start the copy shortcuts operation.

   A dialog box displays showing the corresponding family name and stating that the copy or modify operation is in progress. When the operation is completed, the dialog box closes automatically.

NOTES

- If any error occurs during the copy shortcuts operation, a dialog box displays allowing you to display the log file.
- If there are any unresolved entities on the target side, the browser automatically displays these entities so that you can resolve them manually.

Copy objects in an across model family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Click **Create New Across Model Family**  , and create a copy across model family. For information on how to create a across model family, see *Create a copy across models family using a single grid plane* (on page 1263) and *Create a copy across models family using multiple grid planes* (on page 1264).
3. Select the **Copy Across Model Family** from the **Copy Across Families**  hierarchy.
4. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

5. Click **Create target group copy**  .
6. Click **Finish** to copy the objects from the source ship to the target ship.

Find and correlate objects across models

1. In the target ship, click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Click **Create New Across Model Family** , and create a copy across model family. For information on how to create a across model family, see *Create a copy across models family using a single grid plane* (on page 1263) and *Create a copy across models family using multiple grid planes* (on page 1264).

3. Select the **Copy Across Model Family** from the **Copy Across Families** hierarchy.

4. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

5. Click **Find and correlate** .

   The objects which can be mapped are identified.

6. Click **Finish** to correlate the objects to the target ship from the specified source ship.

   NOTE After you perform **Find and correlate**, the software includes the objects to the copy family so that duplicates are not created during the synchronize operation.

Select the source filter for copy across models

You can perform copy across models without selecting a filter (No Filter) or by specifying the Source Filter.

Performing Copy Across Models without specifying a Source Filter

1. In the target ship, click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the **Copy Across Model Family** from the **Copy Across Families** hierarchy.

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the **No Filter** option.

5. Click **Create target group copy** .

   *The message **As Source Filter is not specified, all the entities in Source Ship will be processed. This could take considerable time. Please press OK to continue or Cancel to stop the operation.***

6. Click **Yes** to perform the copy or **No** to cancel the copy and specify a different filter.

Performing Copy Across Models specifying a Source Filter

1. In the target ship, click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the **Copy Across Model Family** from the **Copy Across Families** hierarchy.
3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays. The **Source Filter** list shows your previously used filters and the **No Filter** and **Select Filter** options. By default, your last used filter is selected. If this is the first time you synchronize the family, **No Filter** is the default.*
4. In the **Source Filter** box, select a filter. To select a filter corresponding to those in the source model of the selected copy across model family, select **Select Filter**. Select a filter from the **Select Filter** dialog box, and click **OK**.

   TIP The **Source Filter** box does not allow you to type a filter name.
5. Click **Create target group copy** .

   If the filter exists, the copy candidates are selected based on the objects associated with the filter.

   TIP If the specified filter does not exist, the message **Filter with the given name does not exist in Source Ship. Please enter the name of an existing Filter** displays. Select **Source Filter for Copy Across Models**.

Schedule a Copy by Family operation as a batch job

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*
2. Click **Batch**.

   *The **Copy by Family Batch** dialog box displays.*

   TIP You must configure the batch queues before you can submit a Copy by Family operation as a batch job. For more information, see *Configure the Smart 3D batch queues* (on page 1274).
3. Set up the parameters to meet your needs. For more information, see *Copy By Family Batch Dialog Box* (on page 1285).
4. Click **Submit**.

   *The **Schedule [Task]** dialog box displays.*
5. Set up the scheduling parameters to meet your needs. For more information, see *Schedule [Task] Dialog Box* (on page 1287).

Configure the Smart 3D batch queues

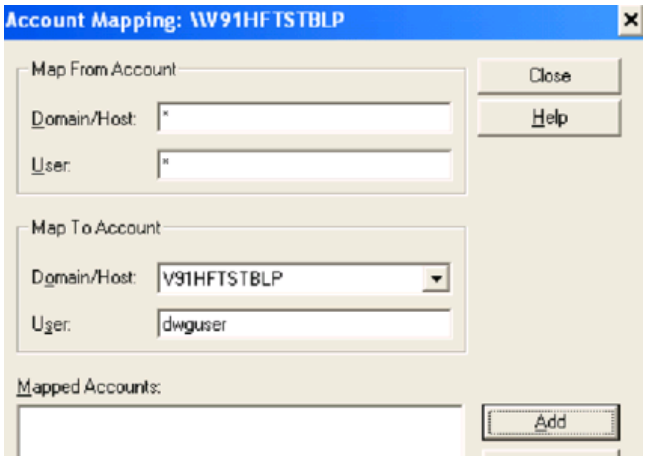
1. Run the configureSP3DBatchQueue.exe utility. By default, this utility is downloaded to the *[Product Folder]*\ProjectMgmt\Tools\Bin folder.

   The software creates, enables, and starts the SP3DProjectManagement, SP3DPrinting, and SP3DApplication queues.

Configure account mapping for batch services

1. Click **Start** > **Intergraph Batch Manager**.
2. Click **Option Server** > **Connect To** to connect to individual servers.
3. Right-click a server, and select **Account Mapping**.

   *The **Account Mapping** dialog box displays.*
4. Type the account mapping details. In the following example, all users and domains from **Map From Account** are mapped to the **Domain/Host: V91HFTSTBLP** and **User: dwguser** account.



5. Click **Add**.
6. Type the password to complete the mapping.

NOTE Other mapping options are also possible. For example, the jobs submitted by User-1 can be mapped to Account-1 while the jobs submitted by User-2 can be mapped to Account-2.

Synchronize Options Dialog Box

Find Options

Clear relationships for Source Filter before Find

Removes any relationships created in the family after clicking **Find and correlate objects**. This restores highlighting of objects in the graphics view for the **Find and correlate objects** step. **Clear then Find** performs the same function as **Refresh Family** on the **Select Copy Family** dialog box.

CAUTION This option removes all **Copy By Family** relationships in the current family for the objects in the source filter.

To In/Outboard

Changes all port and starboard molded conventions to inboard and outboard before using the **Find and correlate objects** command.

Manual Mode

Manually find objects that were not found by the software when using the **Find and correlate objects** command. You can select an object for the **Source Group** and an object for the **Target Group** to make them similar or symmetrical.

Create/Modify Options

Molded Forms objects only

Selects molded forms objects.

Select All

Selects all of the objects.

Select Child

Selects the child objects.

Modify Only Options

Geometry Modifications

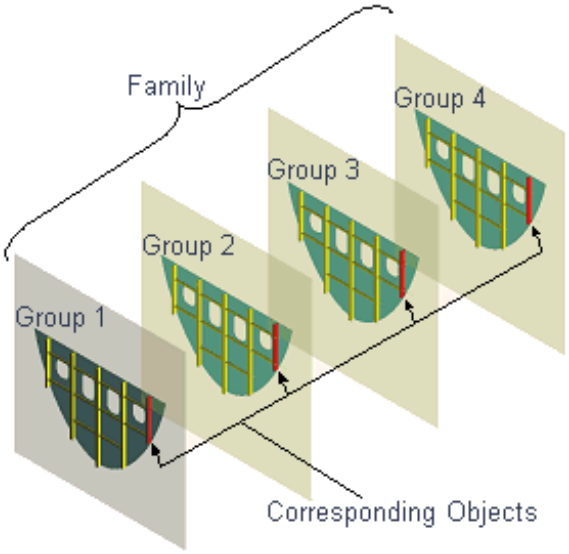
Finds changes in landing curves.

Similar Family Properties Dialog Box

Specifies the groups and grid planes for a Copy Similar family.

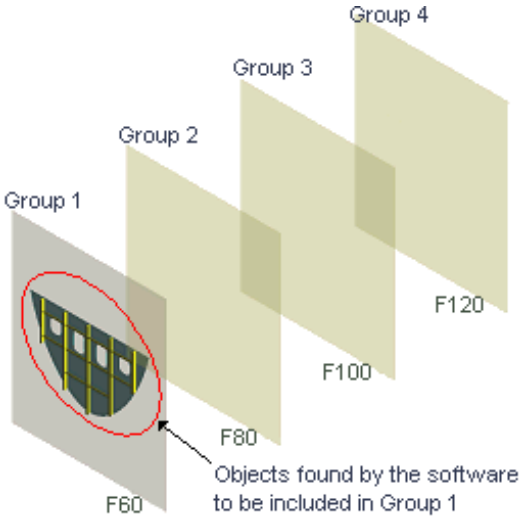
Overview

The Copy Similar method uses a user-defined family of similar grid planes to define what structural objects are copied and to maintain the relationships between the objects to check for modifications. You define a family by selecting the grid planes for each group in the family.



In the following example, Group1 is the source group. The software finds all plate systems that use the Group1 transverse grid plane as the surface definition. The software then finds all

children of the plate system (seams, stiffeners, openings, and features). Finally, the software finds all the objects that are bounded by either the select grid plane or the plate systems defined by the grid plane and all their children.

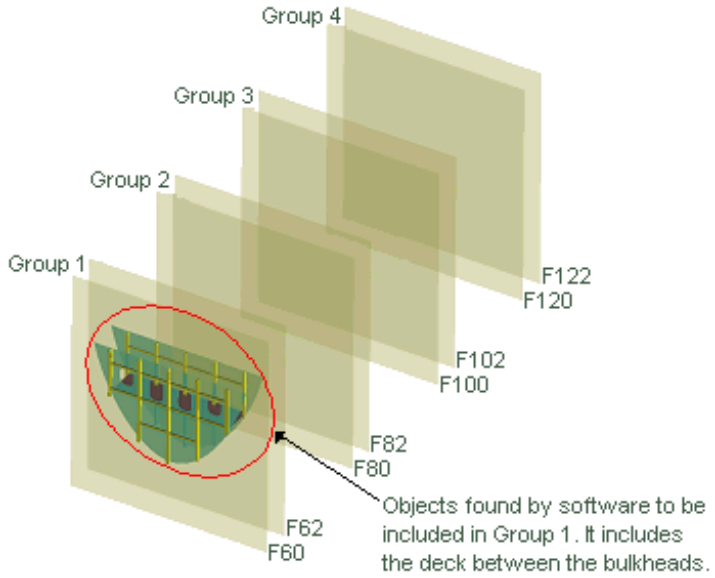


You can use one grid plane or multiple grid planes in each group to define the family.

In this example, one grid plane defines each group, with Group1 being the source group to be cloned for the family.

.G 1 ☑	.G 2 ☐	.G 3 ☐	.G 4 ☐
F60	F80	F100	F120

In the next example, two transverse grid planes define each group, with Group1 being the source group to be cloned for the family. Objects between the grid planes, but related to the grid planes, are also included.



.G 1 ☑	.G 2 ☐	.G 3 ☐	.G 4 ☐
F60	F80	F100	F120
F62	F82	F102	F122

Dialog Box Controls

Name

Type a name for the family. This name must be unique among all family names.

Standard

Defines the Copy Similar Family by specifying one or more rows of grid planes to be used for the groups. For example:

.G 1 ☑	.G 2 ☐	.G 3 ☐	.G 4 ☐
F60	F80	F100	F120
F61	F81	F101	F121
F62	F82	F102	F122
F63	F83	F103	F123
F64	F84	F104	F124

By Range

Defines the Copy Similar Family by specifying a range of grid planes. All grid planes within a range are included. In this example, the results are the same as the five rows specified for the **Standard** example:

.G 1 ☑	.G 2 ☐	.G 3 ☐	.G 4 ☐
F60	F80	F100	F120
F64	F84	F104	F124

Overlapping

Defines the Copy Similar Family by specifying a range of grid planes, but ignores structure that overlaps from one group to the next. For example, in the both the **Standard** and **By Range** examples, a transverse bulkhead could be at the end of the F60-F61 range, but could also be at the beginning of the F61-F62 range. The **Overlapping** option prevents the F61 bulkhead from being created twice.

.G X

Displays the groups that you have defined for this family. Check the corresponding box for the group that you want to copy to the other groups. To add a group to the family, click **Insert group**. To remove a group from the family, click **Remove group**. The name of the group can be modified. For example, **.G 1** can be changed to **Group 1** or **F60 - 62 Group**.

Insert group

Adds a new group to the family.

Remove group

Removes the selected group from the family.

Insert corresponding item (row)

Inserts another row. Click this button when you want to define a group using multiple grid planes.

See Also

Copy by Family (on page 1255)
Copy By Family Log Files (on page 1288)
Create a copy similar family using a single grid plane (on page 1260)
Create a copy across models family using multiple grid planes (on page 1264)
Delete a copy similar family (on page 1262)
Modify a copy similar family (on page 1262)
Select Copy Family Dialog Box (on page 1283)

Symmetry Family Properties Dialog Box

Specifies the groups and symmetry planes for a Copy Symmetry family.

For each symmetry family, a coordinate system is selected, typically the main global coordinate system. The Y=0 grid plane, the longitudinal centerline of the coordinate system, is automatically used by the software as the symmetry plane. The software automatically creates two symmetrical groups, one on each side (port and starboard) of the symmetry plane, containing all the grid planes for each side. One group is selected as the source group to be cloned. For example, the software uses all port side grid planes as the source group. The starboard side grid planes are the corresponding elements for the starboard group. To determine all the structural objects that belong to the source group, the software finds all objects whose definition is based on the grid planes, then finds all their children, then finds all the objects that are bounded by either the grid planes or the systems defined by the grid planes and all their children.

Main tab

Name

Type a name for the copy symmetry family. This name must be unique among all family names.

Symmetry Planes

Select the coordinate systems to use for the symmetry plane. By default, the software uses the Y=0 plane of the selected coordinate system as the symmetry plane. If you select more than one coordinate system, all selected coordinate systems must have coincident Y=0 planes.

Source Group

Select which group is the source for the symmetrical copy.

Port Group Name

Type a name for the port symmetrical group. This name appears when you synchronize the groups. The default name is **Port**.

Starboard Group Name

Type a name for the starboard symmetrical group. This name appears when you synchronize the groups. The default name is **Starboard**.

Range Growth Value

Type a value for the range required for selecting the Copy candidates.

Copy MfgData

Copies manufacturing objects.

Configuration tab

Displays the creation, modification, and status information about an object.

Ship

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which object belongs. You can select another permission group, if needed. Permission groups are created in the Project Management task.

Status

Specifies the current status of the selected object or filter. Depending on your access levels, you may not be able to change the status of the object.

Created

Displays the date and time that the object was created.

Created by

Displays the user name of the person who modified the object.

Modified

Displays the date and time when the object was modified.

Modified by

Displays the user name of the person who modified the object.

See Also

Copy by Family (on page 1255)
Copy By Family Log Files (on page 1288)
Copy Objects in a Symmetry Family (on page 1269)
Create a copy symmetry family (on page 1262)
Delete a copy symmetry family (on page 1263)
Delete objects in a symmetry family (on page 1270)
Modify a copy Symmetry Family (on page 1263)
Select Copy Family Dialog Box (on page 1283)
Similar Family Properties Dialog Box (on page 1276)
Update Objects in a Symmetry Family (on page 1269)

Across Model Family Definition Dialog Box

Defines an Across Model family. An Across Model family is used to perform Copy Similar across ships.

IMPORTANT To use this functionality when your target and source ship are on different servers, you must link the servers. For more information, see *Linking Servers* in the *Intergraph Smart™ 3D Installation Guide*.

Click **Create New Across Model Family** to open the dialog box.

Main Tab

Name

Type a name for the across model family. This name must be unique among all family names. The name **New Across Model Family** is available by default.

Range Growth Value

Type a value for the range required when selecting the Copy candidates. The **Range Growth Value i**s set to **0.80** m by default.

Copy MfgData

Copies manufacturing objects.

Method

Select the method of the definition of the family. You can select either by **Range** or **Standard**. The **Standard** method is selected by default.

Ship Name

Displays the name of the target or source ship. The target ship name (name of the current/active ship) cannot be edited. You can select ships within a site, even if they belong to different servers.

Group Name

Specifies the name of the target/source group. By default, the name of target/source group is the same as the target/source ship name. The target/source group name can be edited.

Corresponding Elements 1

Type a frame name, or browse and select from the list in the **Select Frame** dialog box.

NOTE If the frame name you type is not valid or not available, it is highlighted in red. If you type a new corresponding element name, use the following format:

Coordinate System Name:Frame name. For example, *CS_0:F0* where CS_0 is the Coordinate System name and F0 is the frame name.

Add Corresponding Element

Adds a row for you to type or select corresponding elements.

NOTE Because there are only two corresponding elements allowed when the method definition is **By Range**, the **Add corresponding Elements** button is not available.

Delete Corresponding Element

Removes an element that you have added. This command is available only after you add corresponding elements.

NOTE When the method definition is **By Range**, the **Delete corresponding Elements** button is not available.

See Also

Create a copy across models family using a single grid plane (on page 1263)
Create a copy across models family using multiple grid planes (on page 1264)
Create a copy across models family by range (on page 1264)

Select Filter Dialog Box

Creates, edits, deletes, and selects filters for use with the **Define Workspace**, **Surface Style Rules**, and other **Select by Filter** commands, including Project Management's **Model Data Reuse** (MDR), Drawings View Styles, and Reports commands that require runtime filter selection. You can access this dialog box in several ways.

- Select **File** > **Define Workspace**, and select the **More** option in the **Filter** box.
- Select **Format** > **Surface Style Rules**, click **New** or **Modify**, and then select the **More** option in the **Filter** box.
- Select **Tools** > **Select by Filter.**

The tree view displays the following types of filters:

- **Catalog Filters** - These filters are used to reference data in the Catalog. For example, a catalog filter could apply to company-wide operations. Your administrator can define Company_Filter_1, Company_Filter_2, and so forth.
- **Model Filters** - These filters are available to everyone assigned to a specific model database. There are delivered catalog filters to query on the different types of model objects. You must have the appropriate privileges to create, edit, or delete these filters.
- **My Filters** - These are personal filters that you create and place in the **My Filters** folder. They are visible only to you, the owner. You cannot see the personal filters of others, and they cannot see your personal filters. Select a filter from one of the listed filters, or create a new filter to meet your specific requirements.

 **New Folder**

Creates a new folder.

 **New Filter (Simple or Asking)**

Displays the **New Filter Properties** dialog box so that you can create a new filter. Asking filters allow you to specify the parameters of the search. An asking filter has built-in functionality to ask for values (with boxes that you are required to supply). The values apply to properties that you have already designated you will supply when the filter runs. Asking filters are portable between models.

NOTE Model Data Reuse (MDR) does not support asking filters. The only valid filter types for an MDR transaction are System, Permission Group, Object Type, Volume and Properties. You can define the filter on any one of these tabs or in a combination using multiple tabs.

New Compound Filter

Displays the **New Compound Filter Properties** dialog box, which you use to create a new compound filter containing the Or, And, or Not operators. Compound filters are not supported for MDR.

New SQL Filter

Displays the **New SQL Filter Properties** dialog box, in which you can type the text of an SQL query. SQL filters are not supported for MDR

Delete

Removes a filter or folder from the **Select Filter** list. If you delete a folder, the software also deletes its contents.

Rename

Changes the name of an existing filter or folder from the **Select Filter** list.

Properties

Displays the **Filter Properties** dialog box so that you can select the properties that determine your filter search criteria.

NOTES

- If this dialog box is activated from the **Select by Filter** command, you can select multiple filters on this dialog box. Hold CTRL or SHIFT, and click each filter. When you click **OK**, all objects that fit the selected filters are selected.
- If this dialog box is activated from the **Select by Filter** command, it clears the select set before adding objects to the select set.

See Also

Select by Filter
Select Objects by Filter

Select Copy Family Dialog Box

Toolbar

Create New Similar Family

Creates a new copy similar family. For more information on creating copy similar families, see *Similar Family Properties Dialog Box* (on page 1276).

Create New Symmetry Family

Creates a new symmetry family. For more information on creating copy symmetry families, see *Symmetry Family Properties Dialog Box* (on page 1279).

Create New Across Model Family

Creates a new across model family. For more information on creating across model families, see *Across Model Family Definition Dialog Box* (on page 1281).

Delete Family Definition

Deletes the selected family.

Rename Family Definition

Renames the selected family. You must select a family before you can rename that family.

Edit Family Definition

Edits the selected family. You must select a family before you can edit that family.

Refresh Family

Removes any relationships created in the family by using the **Synchronize** command and the **Copy by Family** ribbon. For example, **Refresh Family** restores highlighting of objects in the graphics view for the **Find and correlate objects** step if that step has been previously carried out.

Copy Shortcuts

Configures copy shortcut keys for similar or symmetry families and specify a log file to store the copy operation. For more information, see *Copy Shortcuts Settings Dialog Box* (on page 1284).

Copy Family Hierarchy

Copy Across Model Families

Lists the Copy Across Model Families that you have defined.

Copy Similar Families

Lists the copy similar families that you have defined.

Copy Symmetry Families

Lists the copy symmetry families that you have defined.

Batch

Displays the **Copy by Family Batch** dialog box. You can use this dialog box to execute jobs in a batch queue. For more information, see *Copy By Family Batch Dialog Box* (on page 1285).

Synchronize

Opens the **Copy by Family** ribbon for the selected family.

Close

Closes the dialog box.

Copy Shortcuts Settings Dialog Box

Configures shortcut keys for similar or symmetry families, and specifies a log file for the copy shortcuts operation.

SNo

Displays the number assigned to the shortcut key.

ShortCut Key

Displays the shortcut key defined in the CopybyFamilyShortCutKeys.txt file. This text file is located in the *[Product Folder]*\CopybyFamily\Data folder.

Family

Indicates whether to copy similar families or symmetry families.

Source Group

Specifies the family group to use as the copy source.

Target Group

Specifies the family group to create or modify.

Select Child

Determines whether to include child objects in the copy operation. Specify **True** or **False** for the value. The default value comes from the **Copy by Family** ribbon.

Molded Form Only

Determines whether the shortcut key applies only in the **Molded Forms** task. Specify **True** or **False** for the value. The default value comes from the **Copy by Family** ribbon.

Log File Path

Specifies a log file for storing the last copy shortcuts operation.

Browse

Click to browse to the log file.

Open

Opens the log file.

OK

Accepts the configuration.

Cancel

Rejects the configuration and closes the dialog box.

See Also

Copy by Family (on page 1255)
Copy By Family Log Files (on page 1288)
Copy objects using Copy Shortcuts (on page 1271)
Select Copy Family Dialog Box (on page 1283)

Copy By Family Batch Dialog Box

Controls properties for copy by family batch processes.

Actions To Queue

Displays the actions in the queue. The options in the grid behave in the same way as those on the Copy by Family ribbon. For more information on these options, see *Copy by Family* (on page 1255).

Family

Specifies the family on which to perform the batch operation. After you select a value for **Family**, the software invokes the source and target groups and source filter.

Steps

Specifies the operation to perform. The list contains the following options:

- **Find**
- **Create**
- **Modify** - This option is not available for copy across model.
- **Delete** - This option is not available for copy across model.
- **Object Report**
- **Property Report**

Source Group

Displays the source group. The software determines this based on the value in the **Family** box.

Target Groups

Displays the target groups. The software determines this based on the value in the **Family** box.

Source Filter

Displays the source filter. The software determines this based on the value in the **Family** box.

Properties

Displays the **Properties** dialog box. For more information, see *Properties Dialog Box* (on page 1288). This option is not available if the **Steps** box is set to **Object Report** or **Property Report**.

Add Action

Adds a row to the end of the grid.

Delete Action

Removes the selected row from the grid. This is the same as pressing DELETE.

Save CBF Log File

Displays the location for the Copy by Family log file. You can type the path, or click **Browse** to navigate to the folder.

Audit Report Path

Displays the location for the Copy by Family audit report file. You can type the path, or click **Browse** to navigate to the folder.

Submit

Displays the **Schedule [Task]** dialog box. For more information see *Schedule [Task] Dialog Box* (on page 1287)

Schedule [Task] Dialog Box

Queue

Displays the name of the queues configured by an administrator for the job. For more information on configuring the queues, see *Configure Queues for Jobs* in the *Project Management User's Guide*.

Run job

Sets the frequency with which the job runs. Jobs can be scheduled to run once or on a regular interval (daily, weekly, or monthly). Depending on the job frequency selected, additional controls display. These controls allow you to define more specific scheduling information. The scheduling controls can be changed only at job submission.

Run on

Sets the time to start running the job.

Options

Opens the Optional Schedule Properties Dialog Box that you can use to define a start and end date.

Run on box

Contains a calendar from which you can select the run date. This option is available when you select **Once** from **Run job**.

Every X days

Specifies how many days pass between job runs. This option is available when you select **Daily** from **Run job**.

Every X weeks

Specifies how many weeks pass between job runs. In addition, you can select on which days the job runs. This option is available when you select **Weekly** from **Run job**.

Day X of the month

Specifies on which day of the month the job runs. This option is available when you select **Monthly** from **Run job**.

The X Y of the month

Specifies on which day of the month the job runs. For example, you can select the last Monday of the month. This option is available when you select **Monthly** from **Run job**.

Job Start

Notifies you when the job starts, if Outlook is set up.

Job Completion

Notifies when the job completes, if Outlook is set up.

Job Abort

Notifies you if the job aborts, if Outlook is set up.

Address Book

Selects the name of the person to be notified by e-mail of the job status, if Outlook is set up. If

Outlook is not available, this option does not work. You can also type the address manually. The person you define here receives an email with the job log files after the job finishes.

NOTES

- The Batch Services SMTP option must be configured on the batch server for this to work. For more information, see the Intergraph Smart Batch Services documentation.
- The WinZip application is no longer required on the batch server to compress any emailed attachments. Compression is now done with functionality included in Smart 3D.

Properties Dialog Box

Controls properties for Copy by Family batch operations.

Clear relationships for Source Filter before Find

Removes all Copy by Family relationships in the current family for the object in the source filter.

Molded Form Objects Only

Indicates that only Molded Forms objects are used for create and modify batch operations.

Select All

Indicates that all objects are used for create and modify batch operations.

Select Child

Indicates that child objects are used for create and modify batch operations.

Geometry Modifications

Indicates that geometry modifications are included in modify batch operations.

Copy By Family Log Files

Logs are generated for **Find and correlate objects**  , **Create target group copy**  , **Modify target group copy**  , and **Finish** operations.

Find and correlate objects  log

The purpose of this log is to show objects that failed to find operations because the target area contains duplicated objects (both geometrically and topologically). You must delete one of the duplicates before the **Find and correlate objects** command can correlate the source and target objects and any dependent objects.

NOTE If the **Modify Port/Starboard** option is selected, the list of forced objects is logged at the finish of the **Find and correlate objects** command.

Create target group copy  log

The initial step of this command shows the candidates for copy (that is, objects that do not yet have a target object). Any objects that are on the to-do list are not selected for copy. None of the dependents of the to-do list objects are selected for copy.

The list of objects on the to-do list is logged, and any dependent objects are listed at the end.

Modify target group copy log

Modify target group copy excludes to-do list objects and reports on them in the same way as the **Create target group copy** command does. **Modify target group copy** compares the attribution of the source object with the target object and selects candidates to copy when differences are found. To simplify the selection process, these differences are reported in the log.

The actual attribute difference is shown first with the objects that are different shown at the end.

Because features are considered attributes of parts, this creates a hierarchy. All features, including intermediate features, are shown.

Finish after Create, Modify, or Delete log

Errors in the finish log include:

- New errors, such as targets that go on the to-do list
- Exceptions thrown by the underlying operations such as copy, split, or detail.
- Exceptions thrown by **Copy by Family** because it is processing an object before its parent. This requires that the copy be run a second time.

The format of these error messages is as follows:

1	!Molded Form target object in error A-614DCK->A-2602DCK/Plate !See Copy Error Management Workflow in Copy By Family User Aid. !Split or Part target object in error A-614DCK_PART->A-2602DCK_PART / Part !See Copy Error Management Workflow in Copy By Family User Aid.
2	Copying: H170P-B27BP-1P -type: Solid !!!Exception processing above elements !!!These elements fail to process, repair and rerun !!!Report error to Intergraph
3	!Skipped DS1(2nd Dk)-491DCK/Plate !Note: Above elements have been skipped, run copy a second time

If the target geometry is bad, the dependents are copied unless they cannot be copied without the parent. The dependents that are not copied are listed at the end of the log.

See Also

Copy by Family (on page 1255)
Select Copy Family Dialog Box (on page 1283)
Copy By Family Log File Workflows (on page 1289)
Similar Family Properties Dialog Box (on page 1276)
Symmetry Family Properties Dialog Box (on page 1279)

Copy By Family Log File Workflows

Target is on the To-Do List

If the **"!Molded Form target object in error A-614DCK->A- 2602DCK/Plate"** type of error is the first error in the log file, check the unsupported object construction list.

- If the source object's construction is not supported by copy, try to fix the target object manually or recreate it manually on the target side.

- If the source object's construction is supported by copy, select the **Clear Before** check box, run **Find and correlate objects**, and try copying again.

After the target object is repaired or created, select the **Clear Before** check box, run **Find and correlate objects** and **Create/Modify/Delete** again to copy any remaining objects.

If the **"!Split or Part target object in error A- 614DCK_PART->A-2602DCK_PART/Part"** type of error is the first error in the log file, check the unsupported object construction list.

- If the source object's construction is not supported by copy, try to fix the target object manually by undetailing and redetailing the parts or by fixing the molded forms system and boundaries. Alternately, you can recreate the object manually on the target side.
- If the source object is supported by copy, contact Intergraph Support, and model the failed target object manually.

After the target object is repaired or created, select the **Clear Before** check box, run **Find and correlate objects** and **Create/Modify/Delete** again to copy any remaining objects.

Exception

If the **"!!!Exception processing above elements"** type of error is the first error in the log file, do the following:

- If the failed object is a Molded Forms object, select the **Clear Before** checkbox, run **Find and correlate objects** again, and then try copying again.
- If the failed object is a detailed part, or if the second try to copy is unsuccessful, contact Intergraph Support, and model the failed target object manually. After the object is repaired or created, run **Find and correlate objects** and **Create/Modify/Delete** again to copy any remaining objects.

Object Skipped

If the **"!Skipped D51(2nd Dk)-491DCK/Plate"** type of error is found in the log file, then there was a problem with the order in which the objects were copied. If you run copy again, the objects are copied correctly, and errors are eliminated from the log.

If errors remain, then there was a problem with the parent of the failed object specified in the skipped error log. Fix the parent so that this object can be copied successfully.

Entities not Completely Supported by the Copy Process

Limitations of the copy implementation can be classified in the following types:

- Object types not yet supported by copy similar/symmetry. The objects are excluded from the copy process. That is, they are not selected as candidates. In some cases, dependent entities might be selected for copy and could fail to copy. These specific cases are listed below.
- Some aspect of the construction of the object is not supported. In this case, you must edit the object to remove the unsupported aspect. These specific constructions are listed below.

Object Types Not Supported By Copy Similar/Symmetry

- Auxiliary coordinate systems. Dependent objects fail.
- Manual logical connections.
- Inverted connections.

- Beam parts (detailed property and beams using assoc points on edges).
- Standalone Plate parts, Profile Parts, Lapped Plate Parts, Bracket Parts.
- Standalone Profile Parts, Profile Edge Reinforcement Parts.
- Straking Seams.
- Profile Straking Seams.
- Split Physical Connections.
- Manufacturing Plates and Profiles.

Aspects of the Construction of Object Not Supported

- Problem with profile and seams by table.
- RAD Sketches containing constraints. After copy, the software replaces the inputs to the sketcher with the new copied geometry. If the new geometry is not infinite, and if the direction of the curves is reversed, this produces bad results for end point constraints and any constraints dependent on the direction. **Copy Symmetry** with edge constraints is not supported.
- Assoc point on edge. Using the place point on edge **Smart Point** placement can result in the point reversal during mirror.

See Also

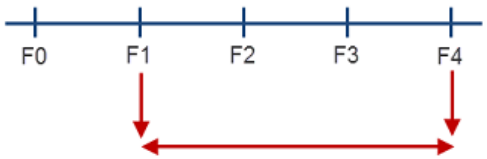
Copy By Family Log Files (on page 1288)

Frame mapping and Range Growth Values in Copy by Family

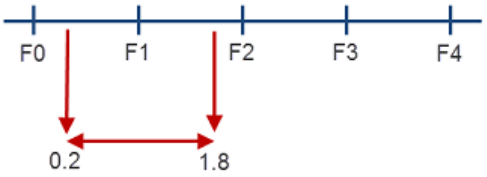
The **Range Growth Value** and **Offset** values determine the range of the family from which you must pick the copy candidates. The technique for using **Range Growth Value** depends on the type of family with which you are working.

Similar Standard Family

If you create a similar standard family with F1 as the source and F4 as the target, the software maps F1 to F4.

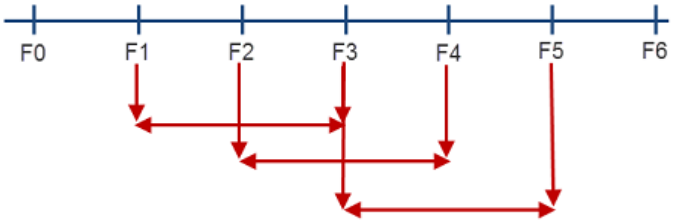


If you specify a default **Range Growth Value** of 0.8, then the range of the family extends from the source frame -0.8 to the source frame + 0.8

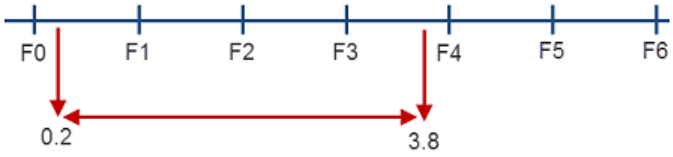


Similar Overlapping Family

To create a similar overlapping family you must have at least one frame in common between the source and the target. For example, you could create a family with F1 and F3 as the source and F3 and F5 as the target. In this case, F3 is the common frame. The software then maps F1 to F3, F2 to F4, and F3 to F5.

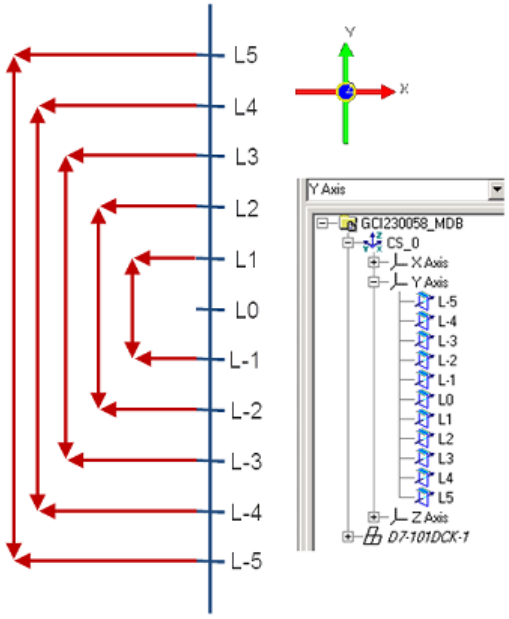


If you specify a default **Range Growth Value** of 0.8, Smart 3D sets the range of the family from the minimum source frame value - 0.8 to the maximum source frame value + 0.8

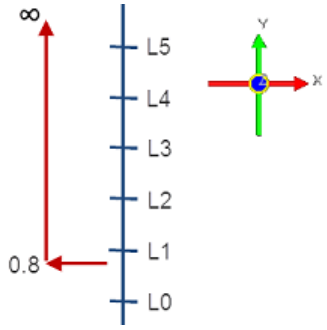


Symmetry Family

If you create symmetry with port as the source with five positive frames and starboard as the target with five negative frames, Smart 3D maps L-1 to L1, L2 to L-2, L -3 to L3, L-4 to L4, and L-5 to L5.

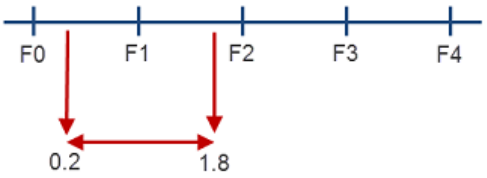


If you specify a default **Range Growth Value** of 0.8, Smart 3D sets the range of the symmetry family from the centerline + 0.8 to infinity.



Across Models Standard Family

If you create an Across-Models Standard Family with F1 as the source and F4 as the target, Smart 3D maps F1 to F4. This behavior is the same as creating a standard family within a model. The software also maps the X and Y axes in an Across Models Family. If you specify a default Range Growth Value of 0.8, Smart 3D sets the range of the family from the source minimum frame value - 0.8 to the source maximum frame value + 0.8.



Objects and behaviors supported for Copy by Family

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
Molded Forms	Planar Plate ▪ Using plane created by **Plane by Vector Normal** ▪ Using plane created by **Plane By Point Normal** ▪ Using plane created by **Plane By 3 Points**	Yes	Yes	Yes
	LEP	Yes	Yes	Yes
	NLEP	Yes	Yes	Yes
	Advanced Plate System ▪ APS polyline plate macros ▪ APS corrugated plate macros ▪ APS swage plate macros	Yes	Yes	Yes
	Bracket by Plane ▪ Using plane created by coincident plane ▪ Using plane created by **Plane By Vector Normal** ▪ Using plane created by **Plane By 3 Points**	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Bracket by selected supports	Yes	Yes	Yes
	Profile by intersection	Yes	Yes	Yes
	Profile by projection	Yes	Yes	Yes
	Profile by geometric construction	Yes	Yes	Yes
	Profile by offset	Yes	Yes	Yes
	Profile by table	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Tripping stiffener	Yes	Yes	Yes
	Profile edge reinforcement	Yes	Yes	Yes
	Seam by intersection	Yes	Yes	Yes
	Seam by offset	Yes	Yes	Yes
	Seam by table	Yes	Yes	Yes
	Seam by geometric construction	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Profile seam	Yes	Yes	Yes
	Openings	Yes	Yes	Yes
	Macro or Catalog openings	Yes	Yes	Yes
	Standard Members	Yes	Yes	Yes
	Built-up (Designed) Members	Yes	Yes	Yes
	Generated logical connections	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Split logical connection having inverted regions	Yes	Yes	Yes
	Manual logical connection	Yes	Yes	Yes
	Plate knuckles	Yes	Yes	Yes
	Profile knuckles	Yes	Yes	Yes
NOTE **Copy by Family** also supports brackets created with **Tripping Bracket**.				
Structural Detailing	Standalone plate part	Yes	Yes	No
	Lapped plate part	Yes	Yes	No

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Plate part	Yes	Yes	No
	Bracket part	Yes	Yes	No
	Profile part	Yes	Yes	No
	Edge features	Yes	Yes	No
	Corner features	Yes	Yes	No
	Sketch features	Yes	Yes	No

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Edge treatments	Yes	Yes	No
	Manual chamfer under edge reinforcement	Yes	Yes	No
	Generated chamfer	Yes	Yes	No
	Smart plate	Yes	Yes	No
	Assembly connections	Yes	Yes	No
	Physical connections	Yes	Yes	No

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	End cuts	Yes	Yes	No
	Slot	Yes	Yes	No
	Collar	Yes	Yes	No
Structural Manufacturing	Constant margin	Yes	Yes	No
	Oblique margin	Yes	Yes	No
	Marking	Yes	Yes	No

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Shrinkage	Yes	Yes	No

Objects not supported by Copy by Family

Task	Objects Not Supported by Copy by Family
Molded Forms	Objects created using **Reference Curve by Intersection**
	Ruled plates
	Child plates
Structural Detailing	Bracket part
	Standalone stiffener part
	Beam profile part
	Plate edge reinforcement part
	Profile edge reinforcement part
	Profile straking seam
	Split physical connection
	Member generic assembly connection
	Manual physical connection
Structural Manufacturing	Template
	Pin Jig
	Manufacturing Plate
	Manufacturing Profile
NOTE No other objects from other tasks (such as Equipment, Hangers and Supports, and so on) are supported in **Copy by Family**.	

Objects and behaviors supported in Attribute Modify

Task	Objects Supported for Modify in Copy by Family	Properties Supported
Molded Forms	Plate	▪ Type ▪ Naming Category ▪ Specification ▪ Continuity ▪ Tightness ▪ Description ▪ Split Priority ▪ Material ▪ Material Grade ▪ Actual Thickness ▪ Plate System Type ▪ Thickness Direction ▪ Offset ▪ Primary Profile Orientation Forward of For Aft reference ▪ Primary Profile Orientation Aft of For Aft reference ▪ Primary Profile Orientation at For Aft reference ▪ Secondary Orientation of Vertical Profiles ▪ Secondary Orientation of Vertical Profiles on Port Starboard ▪ Secondary Orientation of Horizontal Profiles ▪ All user-defined attributes

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Brackets	▪ Type ▪ Naming Category ▪ Specification ▪ Continuity ▪ Tightness ▪ Split Priority ▪ Structural Priority ▪ Material ▪ Material Grade ▪ Actual Thickness ▪ Thickness direction ▪ Offset ▪ Primary Profile Orientation ▪ Primary Profile Orientation at Port Starboard Reference ▪ Secondary Orientation of Longitudinal Profiles ▪ Secondary Orientation of Vertical ▪ Profiles Forward of Fore Aft reference ▪ Secondary Orientation of Vertical ▪ Profiles Aft of Fore Aft reference ▪ Secondary Orientation of Vertical ▪ Profiles at Fore Aft reference ▪ Bracket Contour Type ▪ All user-defined attributes

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Profile	▪ Type ▪ Naming Category ▪ Continuity ▪ Continuity Priority ▪ Description ▪ Section Type ▪ Section Size ▪ Material ▪ Grade ▪ Primary Orientation ▪ Secondary Orientation ▪ Mounting Face ▪ Load Point ▪ Landing Curve Projection Method ▪ Attachment Method ▪ Start Connection Method ▪ End Connection Method ▪ All user-defined attributes

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Profile Edge Reinforcement	▪ Naming Category ▪ Specification ▪ Description ▪ Section Type ▪ Section Size ▪ Material ▪ Grade ▪ Position ▪ Primary Orientation ▪ Secondary Orientation ▪ Mounting Face ▪ Load Point ▪ Start Connection Method ▪ End Connection Method ▪ All user-defined attributes
	Seam	▪ Seam type ▪ All user-defined attributes
	Profile Seam	▪ All user-defined attributes
	Opening	▪ Opening Design Type ▪ Opening Planning Method ▪ Opening Production Method ▪ All user-defined attributes
	Manual/Generated Logical Connections	Only user-defined attributes
	Standard Member	▪ Type ▪ Type Category ▪ Priority ▪ Continuity Type ▪ Continuity Priority number ▪ Align

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Member Part	▪ Type ▪ Type Category ▪ Section Name ▪ Cardinal Point ▪ Angle ▪ Reflect ▪ End Releases ▪ Member Section Name ▪ Material ▪ Grade
	Split Connection	▪ Offset ▪ Offset Reference ▪ Distance ▪ Location Definition
	SPS Opening	▪ Cutting Limit ▪ All user-defined attributes
	Frame Connection	▪ Connection ▪ All user-defined attributes
	Built-up Member	▪ Type ▪ Type Category ▪ Priority ▪ Continuity Type ▪ Continuity Priority number ▪ Align

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Designed Member	▪ Cardinal Point ▪ Angle ▪ Reflect ▪ Type Category ▪ Type ▪ Priority ▪ Length Extension ▪ Wide Extension - Top Flange ▪ Depth Extension
Structural Detailing	Plate Part	▪ Description ▪ Thickening Technique Requested ▪ All user-defined attributes
	Bracket Part	▪ Description ▪ Thickening Technique Requested ▪ All user-defined attributes
	Profile Part	▪ Naming Category ▪ Description ▪ All user-defined attributes
	Edge Feature	Not Supported
	Corner Feature	Not Supported
	Sketch Feature	Not Supported
	Edge Treatment	Not Supported
	Standalone Plate Part	Not Supported

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Assembly Connections	▪ End Cut Type ▪ Split End To End Case ▪ Flip End To End Angle ▪ Chamfer Type ▪ Current Selection ▪ Question Controls ▪ All user-defined attributes
	Physical Connections	All attributes are supported
	Free End Cuts	▪ End Cut Type ▪ Chamfer Type ▪ Current Selection ▪ Question Controls ▪ Is Named By Rule
	Webcut	All attributes are supported
	Flangecut	All attributes are supported
	Smart Plate	▪ Description ▪ Thickening Technique Requested ▪ All user-defined attributes
	Slot	All attributes are supported
	Collar	All attributes are supported
Structural Manufacturing	Margin	▪ Type ▪ Value ▪ Start Value ▪ End Value
	Marking	▪ Manufacturing Marking Side ▪ Manufacturing Marking Type
	Shrinkage	▪ Shrinkage Type ▪ Primary Factor ▪ Secondary Factor

Geometry modification behavior for objects in Copy by Family

Application	Objects Supported for Modify with Geometry Modification in Copy by Family	Remarks
Molded Forms	Plate	Only boundary modifications are supported.
	Brackets	Only boundary modifications are supported.
	Profile	Geometry modification only applies to boundary and landing curve modifications.
	Seam	Geometry modification only applies to boundary and landing curve modifications.
	Seam Point	Geometry modification applies to reference plane and offset cases.
	Openings and Macro Openings	Geometry modification applies to the location change of the opening.

Solutions to common problems in Copy by Family

Problem: A problem is encountered when creating a symmetry family.

Reason: Creating a symmetry family requires an equal number of positive and negative frames for a coordinate system.

Recovery: Update the coordinate system so that there are an equal number of positive and negative frames for the coordinate system.

Problem: The following error message displays in the log file:

```
The Object is being copied before its input <NoBlk>-5WB.101-1-PS.1-1
1. Please Check the Geometry and Inputs for this Object from Source to
Target
2. Try to do Manual Copy of the Parent and Perform Find and Correlate.
Aborted due to Issue with Input Ordering.
       !Skipped A-221LBH-1/Plate
       !Note: Above elements have been skipped, run copy a second time
```

Reason: **Copy by Family** did not handle the ordering and wrote the message to the log file.

Recovery: Use **Copy by Family** multiple times to complete the copy.

OR

Use **Copy/Paste** to create a find to map for these objects.

Problem: Smart 3D does not find an object using **autoFind**.

Reason: The objects are not identical, or the objects are not identical crossovers.

Recovery: Perform a manual **Find** using the browser to map the objects.

Problem: Objects depending on a hull do not copy using **Across Models Family**.

Reason: Either the source or target model has more than one hull.

Recovery: Perform a manual **Find** using the browser to map the hull from the source to the target model.

Problem: The **Range Growth Value** does not impact the selection of objects.

Reason: You have modified the **Range Growth Value** after clicking **Create**.

Recovery: Refresh the family, and then select the appropriate objects.

Problem: Smart 3D does not select objects for **Modify**, even though the objects have differences in attributes and geometry.

Reason: The objects are out of the workspace, or the **Geom-Mod** option is not available on the ribbon.

Recovery: Verify that the source and target objects are in the workspace. Use the **Geom-Mod** option if a change exists in the geometry of the source object.

Problem: Objects dependent on leaf systems or parts fail to copy.

Reason: Smart 3D performs the split operation step which generates leaves on the target side during the split operation step. This step follows the Molded Forms copy step.

OR

Smart 3D performs the detailing step (which creates detailed parts on the target side) after the Molded Forms copy step.

Recovery: Perform multiple runs to copy this type of object.

Frequently Asked Questions

Q: Does Copy by Family support copy of objects created using a rotated coordinate system?

A: No, Copy by Family cannot be used to copy objects if the objects are created using rotated coordinate systems.

Q: How do I verify that the object is copied to the target?

A: Use **Audit-Tool** in **Manual Mode** to select the family and select the object to check from Smart 3D. Then, run a report to show whether Smart 3D copied the object to the target.

Q: How do I map objects that are not identical from the source to the target?

A: Use the browser in **Manual** mode to map the objects.

Q: How do I copy only certain objects in an **Across Models Family**?

A: Create a filter on the source with the objects to copy. Use that filter in the **Source Filter** box so that Smart 3D only processes the required objects.

Q: How do I copy objects that are children of an object I am not copying with **Copy by Family**?

A: Use **Copy/Paste** or **Mirror Copy** to copy the objects to the target. Then, use **Find** to map the object, and **Copy** the child objects using **Copy by Family**.

Q: How do I copy only certain objects from the source?

A: Press CTRL + Q to use the **Manual Preselect** option to manually select the objects to copy and synchronize. Smart 3D then makes **Finish** available. Click **Finish** to copy these objects to the target. This technique also works for **Modify**.

Q: When do I use **Select All** and **Select Child**?

A: To select all the objects, click **Select All**. The selected objects display in yellow.

To clear all the objects so that you can pick the objects to copy, clear **Select All**. The objects then display in cyan. As you select the objects to copy, they display in yellow.

To select all the child objects when you select the parent, click **Select Child**.

Q: Are some objects not supported by the audit tool?

A: The audit tool does not report logical connections created by manually created features copied using **Copy by Family**.

Q: Does **Copy by Family** support all kinds of filters?

A: The command supports ship filters, SQL filters, and user filters. **Copy by Family** does not currently support asking filters**.**

Q: Which kind of coordinate systems can I use to create a symmetry family?

A: Currently, Smart 3D only supports ship coordinate systems for creating a symmetry family.

Q: What should I do if I get an HRESULT failure?

A: HRESULT failures can occur because of compilation or build issues. Try to minimize the test case by reducing the number of objects to determine the object that is causing the HRESULT failure. Then, file a TR with the simplified workflow.

Q: What should I do if I see an **Out of memory** issue during the copy process?

A: Use the **3GB** option and **GC Cache** to copy the objects from the source to the target. File a TR so that the memory issues are resolved.

Q: What is the difference between **Manual Mode** in **Find** and **Manual Find** in the browser?

A: Smart 3D uses both options to map source to target objects. In **Manual Mode**, you can resolve one object at a time. You can use **Manual Find** to resolve multiple objects at a time.

Q: What is the custom filter option in **Copy by Family**?

A: You can select either a workspace filter or any specific filter for performing the following actions:

- **Find/Create/Modify/Delete** option in **Copy by Family**.
- Select objects for processing that are common to the workspace by using the **Custom Filter** option.

Q: What issues do I face when I use coordinate systems with different frame spacing in **Across Models Family**?

A: **Issue 1** - If you create objects using one coordinate system but create the family using another coordinate system, Smart 3D creates substitute planes at an improper location, because the frame spacing is different for both of the coordinate systems. Because of this, use the same coordinate system for both the family and the objects.

Issue 2 - If you create the hull and range family with coordinate systems having different frame spacing, hull mapping fails. This also causes all of the dependents of the hull to fail to copy.

Q: Does **Copy by Family** support modification (**Property** or **Geometry**)?

A: No, you cannot propagate feature modification to the target by using the **Copy by Family** modify operation. You can achieve this by updating the feature count on the source by

adding or removing features on the source. **Copy by Family** picks up this feature count as a difference and deletes and recopies all of the features to the target.

Q: What kind of advanced plate system (APS) plates are supported using **Copy by Family**?

A: **Copy by Family** supports the following advanced plate systems:

- Plate systems created using APS polyline plate macros
- Plate system created using APS corrugated plate macros
- Plate system created using APS swage plate macros

Q. What is the **System Hierarchy** option in **Copy by Family**?

A. Copy by Family provides target parent system options on the **Select Copy Family** dialog box so that the target objects are copied under the given parent system. This makes target objects easier to locate.

Q: What is **Copy/Paste**? How can I use this command in conjunction with **Copy by Family**?

A: **Copy/Paste** lets you copy and paste objects from the source to the target by taking inputs from the **Paste** dialog box.

If an object fails to copy from the source to the target using **Copy by Family**, then all the dependent objects also fail to copy because the parent is missing on the target. Use **Copy/Paste** to copy such objects, and perform an auto or manual **Find** to map the source to the target object. Then, use **Copy by Family** to copy the dependent objects.

Q: What should I do to decide if a problem is related to **Copy by Family**?

A: Perform the following actions:

a. Verify that the object is supported by **Copy by Family**. For more information, see *Objects and behaviors supported for Copy by Family* (on page 1294).

b. If the object is supported, check for a **ToDo** list entry.

c. Try to **Edit** the object.

d. Try to **Copy/Paste** the object.

e. Try **Model Data Reuse** for the object.

If all the previous actions are successful, then the issue is related to **Copy by Family**.

Q: What is the port mapping issue in **Copy by Family**?

A: If the following message displays in the finish log, then this is a port mapping issue in **Copy by Family**.

```
WARNING :  Cannot resolve input mapping for object->
IJPortSrc-SF_!BLFX_0_0_8_<>_{}_()||D3-DCK
!Skipped F 70.000 m-1TBH-1/Plate
!Please review with support
```

Possible reasons:

- Smart 3D did not copy an object that the port needs.
- An ordering issue from **Copy by Family** exists.
- A port mapping issue from **Copy by Family** exists.

Use **Copy/Paste** to copy the object, and then use **Auto** or **Manual Find** to map the objects using **Copy by Family**.

Q: What are the implications of using **Commit** in rules?

A: Avoid using **Commit** and client-tier references in rules. Using these references can cause Smart 3D to exit.

SECTION 26

Copy By Family Synchronize

The **Edit** > **Copy by Family Synchronize** command opens the **Copy by Family** ribbon with the last used family set as the default family.

NOTE This command can be used if you want to perform **Copy by Family** for families that already have been created. If no families exist, a message is displayed informing you that there are "**No Copy Families Defined**".

See Also

Copy By Family Log Files (on page 1288)
Select Copy Family Dialog Box (on page 1283)
Similar Family Properties Dialog Box (on page 1276)
Symmetry Family Properties Dialog Box (on page 1279)

SECTION 27

Copy By Family Audit Report

The **Tools > Copy by Family Audit Report** command allows you verify that the source has been copied to the target by highlighting selected source and targets after a copy has been performed.

Copy By Family Audit Report Ribbon



Edit file report properties

Opens the **Settings** dialog box in which you set the path to the location where you want to save the generated report. By default, the path for all the reports is set to C:\Documents and Settings*[User name]*\Smartmarine3d\Report Output.

NOTE Currently, three kinds of reports are generated. These reports are generated in the **Microsoft Excel** workbook format and can be saved for future reference.

- **Object Summary Report** - Provides information on object details.
- **Property Summary Report** - Verifies that the properties of source and target objects match.
- **Property Compare Report** - Provides a detailed description of the Property Summary Report.



Select Objects

Select objects manually from **Workspace Explorer** or from the graphic view when **Select Mode** list is set to **Manual**.



Run Report

Allows you process all the objects and displays the report. The **Run Report** option is not available if it is **Manual** unless an object is selected manually.

Finish

Closes the reports. This option is only available after you run a report.



Reject

Rejects all selected objects.



Accept

Accepts all selected objects.

Report

Select **Object Summary Report** or **Property Summary Report.** The **Object Summary Report** is available by default.

Select Mode

Lists the following modes:

- **Manual** - Selects objects manually from the **Workspace Explorer** or graphic view.

- **Auto Select Family Objects** - Selects all objects associated with the selected family for processing.
- **Auto Select Family Objects in Workspace** - Selects all objects associated with the selected family which are in current workspace.
- **Open Existing Reports** - Displays existing reports.
- **Manual (recursive)** - This is same as **Manual** mode except that in this case, when an object is selected, its children also get selected.

Copy Family

Select the family on which you want to run the report. Acceptable values are **Copy Across Models Family**, **Similar Family**, or **Symmetry Family**.

Object Summary Report

The **Object Summary Report** has three worksheets.

- Worksheets with object information. These worksheets are named in the following format - “Source group-Target group”.
- **Configuration** worksheet
- **Defective Entities** worksheet

NOTE The report has multiple worksheets for multiple targets.

Worksheet with Object Information

In the worksheet with object information, you can find the following information:

Status

Displays the status of the object. If the object is copied, the status is **OK.** If the object is not copied, the status is **Check**.

NOTE When the status is **Check**, you need to check for any discrepancies in the source and the target. The entire row is highlighted in yellow. The parameters which fail to match are highlighted in red.

Object Type

Displays information on the type of object.

Source Object Name

Displays the name of the source object.

TDL

Indicates whether this object (**Source** or **Target**) is present in the To Do List.

LC's

Displays the count of logical connections.

AC's

Displays the count of assembly connections.

PC's

Displays the count of physical connections.

Fetr's

Displays the number of features available for the object.

Detailed

Displays a detailed status on the object.

Copied

Specifies whether the object has been copied successfully.

Target Object Name

Displays the name of the target object.

Configuration Worksheet

The **Configuration** sheet provides you with the following information:

Generation Date

Displays the date when the report was generated.

UserID

Displays user identification information.

Selected Mode

Specifies the mode selected.

Compute Time

Displays the time taken (in seconds) for generating the report.

Number of Objects

Displays the number of objects selected.

Defective Entities Worksheet

If there is any information about defective entities, it displays on the **Defective Entities** sheet.

NOTE When a cell in any row is selected in the report, the corresponding source and target objects are highlighted in the graphic view.

You can also run the **Object Summary Report** on a **Copy Across Model** family.

Property Summary Report

The **Property Summary Report** has two work sheets - **Parameter Summary** and **Defective Entities**.

In the **Parameter Summary** sheet, you can find the following information:

Status

Displays the status on the properties of the source and target objects. If all the properties for the source and target match, the status is **Ok**. If any of the properties do not match, the status is **Check**.

NOTE If the status is **Check**, the entire row is highlighted in yellow. For multiple targets, the status is **Check** if the properties of the source object fail to match with any of the targets. In this case, the cell corresponding to that target is highlighted in red.

Object Type

Displays information on the type of object selected.

Source

Displays the name of the source object.

Show Details

Describes the **Property Summary Report** in detail**.**

Target

Displays the name of the target object.

NOTE In all the reports, any discrepancies in source and target are highlighted in red.

Property Compare Report

The **Property** sheet lists all of the properties of the source and targets and their status. In the **Property** sheet, you can find the following information:

Status

Displays the status for each of the properties of the source and target objects.

NOTES

- If the status is **Check**, it indicates that the property of the source and target has failed to match. The entire row is highlighted in yellow. If the property matches, the status is **Ok**.
- When the status is **Check**, a comment is displayed on the target box (when you move the pointer over the target box) indicating the expected value that is needed for the properties to match.
- When the target value is expected to be different (in cases, when target attribute value is defined by a rule), the status is **Ok**.

Property

Displays information on any property associated with the selected objects.

Source

Displays the name of the source object.

Target

Displays the name of the target object.

NOTE If the objects being processed for a particular report do not belong to the selected family, then the following message displays: **There are some objects not associated with the family. Hence these objects will not be reported in the Report.**

What do you want to do?

- *Generate an object summary report in manual mode* (on page 1320)
- *Generate an object summary report in auto select family objects mode* (on page 1320)
- *Generate an object summary report in auto select family objects in workspace mode* (on page 1320)
- *Generate a property summary report in manual mode* (on page 1321)

- *Generate a property summary report in auto select family objects mode* (on page 1321)
- *Generate a property summary report in auto select family objects in workspace mode* (on page 1321)

Generate an object summary report in manual mode

You can generate the object summary report in manual mode to generate the report for required objects in family defined in the workspace.

1. Click **Tools > Copy By Family Audit Report**.

   *The **Audit Report** ribbon displays.*

2. In the **Report** box, select **Object Summary Report**.
3. In the **Select Mode** box, select **Manual**.
4. In the **Copy Family** box, select the family on which to run the report.
5. Select the objects from **Workspace Explorer** or the graphic view.
6. Click **Run Report** to display the **Object Summary Report**.

Generate an object summary report in auto select family objects mode

If you want to select all the objects related to the family without regard to the defined workspace (for example, all the related objects in the database), you can generate the report in auto select family objects mode.

1. Click **Tools > Copy By Family Audit Report**.

   *The **Audit Report** ribbon displays.*

2. In the **Report** box, select **Object Summary Report.**
3. In the **Select Mode** box, select **Auto Select Family Objects**.
4. In the **Copy Family** box, select the family on which to run the report.
5. Click **Run Report** to display the **Object Summary Report**.

Generate an object summary report in auto select family objects in workspace mode

You can generate the object summary report in auto select family objects mode to generate the report for required objects in a family defined in the workspace.

1. Click **Tools** > **Copy By Family Audit Report**.

   *the **Audit Report** ribbon displays.*

2. In the **Report** box, select **Object Summary Report**.

3. In the **Select Mode** box, select **Family Objects in Workspace**.
4. In the **Copy Family** box, select the family on which to run the report.
5. Click **Run Report** to display the **Object Summary Report**.

Generate a property summary report in manual mode

You can generate the property summary report in manual mode to generate the report for specific objects in the family defined in the workspace.

1. Click **Tools > Copy By Family Audit Report**.

   *The **Audit Report** ribbon displays.*
2. In the **Report** box, select **Property Summary Report.**
3. In the **Select Mode** box, select **Manual**.
4. In the **Copy Family** box, select the family on which to run the report.
5. Select the objects from **Workspace Explorer** or graphic view.
6. Click **Run Report** to display the **Property Summary Report**.

Generate a property summary report in auto select family objects mode

If you want to select all the objects related to the family without regard to the defined workspace (for example, all the related objects in the database), you can generate the report in **Auto Select Family Objects** mode.

1. Click **Tools > Copy By Family Audit Report**.

   *The **Audit Report** ribbon displays.*
2. In the **Report** box, select **Property Summary Report.**
3. In the **Select Mode** box, select **Auto Select Family Objects**.
4. In the **Copy Family** box, select the family on which to run the report.
5. Click **Run Report**.

   *The **Property Summary Report** displays.*

Generate a property summary report in auto select family objects in workspace mode

You can generate the property summary report in auto select family objects mode to generate the report for required objects in a family defined in the workspace.

1. Click **Tools** > **Copy By Family Audit Report**.

   *The **Audit Report** ribbon displays.*
2. In the **Report** box, select **Property Summary Report**.

3. In the **Select Mode** box, select **Family Objects in Workspace**.
4. In the **Copy Family** box, select the family on which to run the report.
5. Click **Run Report**.

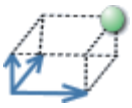
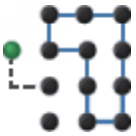
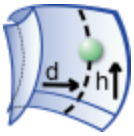
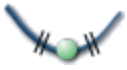
   *The **Property Summary Report** displays.*

SECTION 28

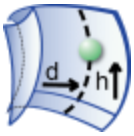
Topological Points (Insert Menu)

Insert > **Topological Points** creates associative points directly in the graphic view. The points parametrically constrain objects that use them. These points are most commonly used with the **Plane by Three Points** plane method used in many commands.

This command opens the **Point Palette** dialog box and contains the following options:

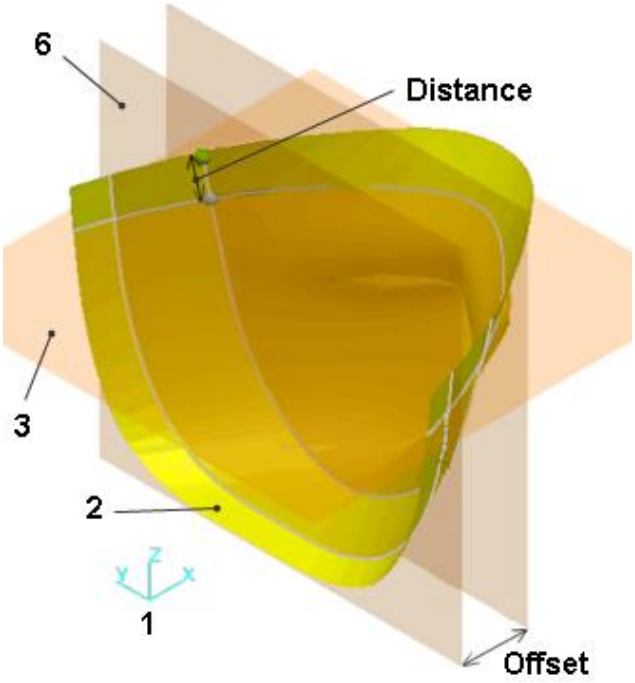
	**Point from Coordinate System** - Creates a point at the defined X, Y, and Z coordinates of the selected coordinate system. For more information, see Point from Coordinate System.
	**Point at Minimum Distance** - Creates a point at defined distances from the faces of two plate, profile, or member systems. For more information, see *Point at Minimum Distance* (on page 1341).
	**Point at Offset from Key Point** - Creates a point using a cross-section key point as a reference. For more information, see *Point at Offset from Key Point* (on page 1348).
	**Point Along Curve on Surface** - Creates a point along a curve relative to the selected coordinate system. For more information, see *Point Along Curve on Surface* (on page 1029).
	**Point at Curve Middle** - Creates a point at the middle of a curve. For more information, see *Point at Curve Middle* (on page 1335).

Point Along Curve on Surface



Tooltip: PointAlongCurveOnSurface

Creates a point along a curve on a surface relative to the selected coordinate system.



Required Input	**Output**
1 - Coordinate system	Point along the curve
2 -Surface	
3 - Reference plane intersecting the surface	
6 - Supporting plane intersecting the surface	
Optional Input	
Points or curves to define reference plane **3**	
Parameters	
Offset method and value	
Distance method and value	

Point Along Curve on Surface Ribbon (for geometric constructions)

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the coordinate system. The coordinate system is used when there is an ambiguous solution.

2

Specifies the surface. By default, this is the surface of the plate selected for the main command.

3

Specifies a reference plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the reference curve from which the new point is offset as defined by **Offset** and **Offset Direction**.

(4)

Specifies a point used to define the plane intersecting the surface. This option is only available if a point or curve is selected for **3**.

(5)

Specifies a third point used to define the plane intersecting the surface. This option is only available if points are selected for **3** and (4).

6

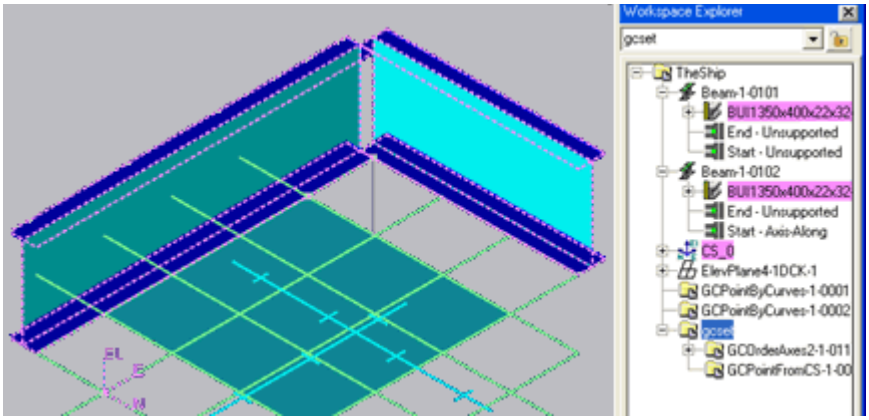
Specifies a supporting plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the supporting curve along which the new point is located as defined by **Distance** and **Distance Method**.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

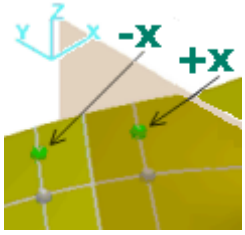
★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Offset

Defines the offset distance of the point along the reference curve.

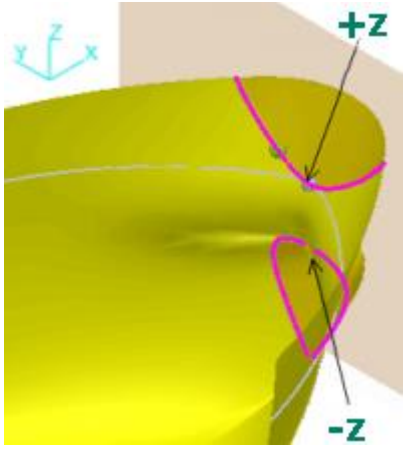
Offset Direction

Defines the offset direction (with respect to the selected coordinate system **1**) along the reference curve. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available when **Offset** is set to a value other than **0**.



Inter Ambiguity

Specifies the solution when the supporting plane intersects a complex surface in multiple locations. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. The following example shows the ambiguity options in the bow of a hull.



Distance

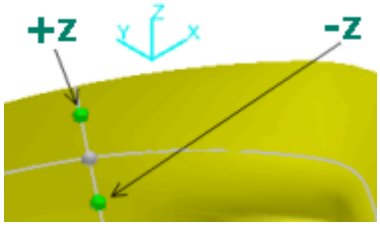
Defines the distance of the point from the reference curve along the supporting curve.

Distance Method

Defines how the point is measured along the supporting curve. Select **Girth** (on page 1457), **Along x**, **Along y**, or **Along z**.

Point Ambiguity

Specifies the solution along the supporting curve when **Distance** is not **0** and **Girth** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Girth**.

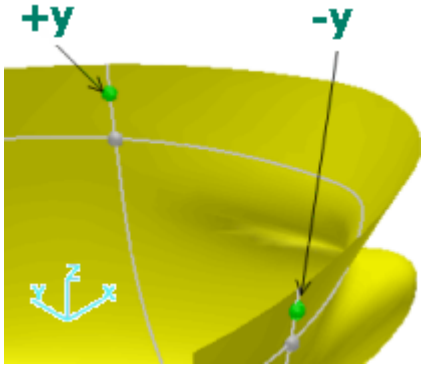


Distance Direction

Specifies the solution along the supporting curve when **Distance** is not **0** and **Along x**, **Along y**, or **Along z** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Along x**, **Along y**, or **Along z**.

Side Ambiguity

Specifies the solution when the reference plane intersects a complex surface in more than one location, such as on the port and starboard sides of a hull. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**.



WARNING For options that have coordinate axis values, select a value that is compatible with the orientation of the curve. For example, if a non-linear reference curve is in the x-y plane, valid options for **Offset Direction** are **+x**, **-x**, **+y**, and **-y**.

Point Along Curve on Surface Ribbon (for topological points in the Molded Forms task)

Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

Specifies the coordinate system. The coordinate system is used when there is an ambiguous solution.

2

Specifies the surface. By default, this is the surface of the plate selected for the main command.

3

Specifies a reference plane intersecting the surface, or a curve or point used to define the plane. The curve created by this intersection is the reference curve from which the new point is offset as defined by **Offset** and **Offset Direction**.



Specifies a point used to define the plane intersecting the surface. This option is only available if a point or curve is selected for **3**.



Specifies a third point used to define the plane intersecting the surface. This option is only available if points are selected for **3** and  .

6

Specifies a supporting plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the supporting curve along which the new point is located as defined by **Distance** and **Distance Method**.

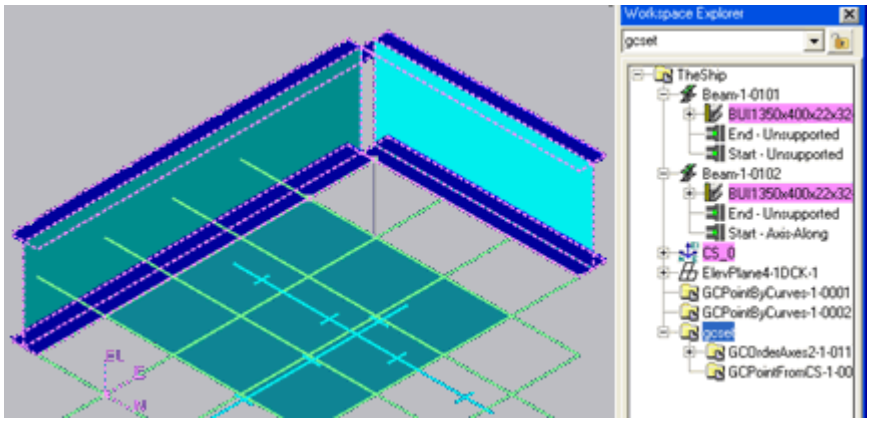
Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

-  - No background when no input is selected.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- 1 and 0.00 m - No background when the value of the original geometric construction is retained.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

- 1 - Blue background when the input is the same for all selected geometric constructions.
- 1 - No background with pink underline when the input is different for *all* selected geometric constructions.
- 1 - Blue background with pink underline when the input is different for *some* selected geometric constructions.
- 1 - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.
- - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.



Reject

Clears the current selection.

Accept

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.
- **More** - Select a system in the model.

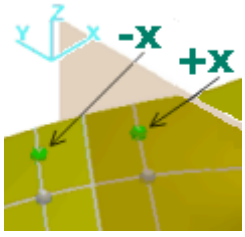
NOTE The **New GC Set** option is not used for topological points.

Offset

Defines the offset distance of the point along the reference curve.

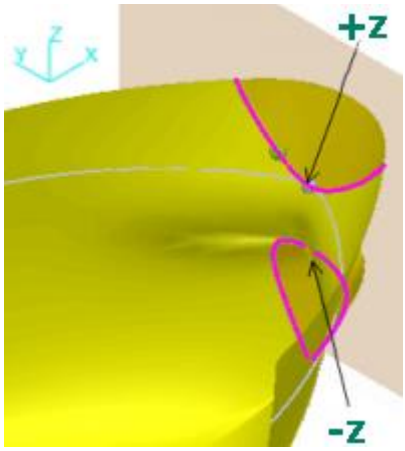
Offset Direction

Defines the offset direction (with respect to the selected coordinate system **1**) along the reference curve. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available when **Offset** is not set to **0**.



Inter Ambiguity

Specifies the solution when the supporting plane intersects a complex surface in multiple locations. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. The following example shows the ambiguity options in the bow of a hull.



Distance

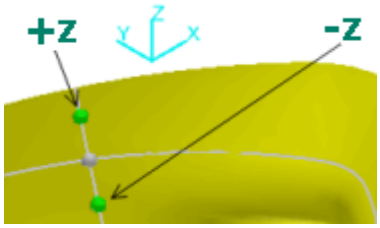
Defines the distance of the point from the reference curve along the supporting curve.

Distance Method

Defines how the point is measured along the supporting curve. Select **Girth** (on page 1457), **Along x**, **Along y**, or **Along z**.

Point Ambiguity

Specifies the solution along the supporting curve when **Distance** is not **0** and **Girth** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Girth**.

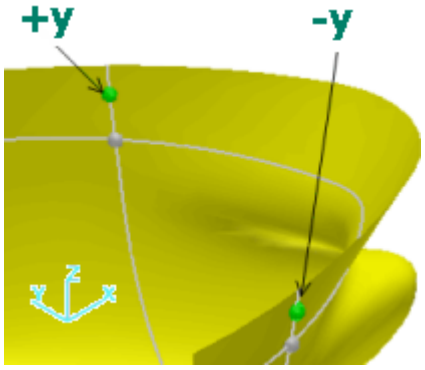


Distance Direction

Specifies the solution along the supporting curve when **Distance** is not **0** and **Along x**, **Along y**, or **Along z** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Along x**, **Along y**, or **Along z**.

Side Ambiguity

Specifies the solution when the reference plane intersects a complex surface in more than one location, such as on the port and starboard sides of a hull. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**.



★ **IMPORTANT** For options that have coordinate axis values, select a value that is compatible with the orientation of the curve. For example, if a non-linear reference curve is in the x-y plane, valid options for **Offset Direction** are **+x**, **-x**, **+y**, and **-y**.

What do you want to do?

- *Place a point on a surface and along a curve* (on page 1038)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)
- *Place a topological point on a surface and along a curve* (on page 1333) (in the Molded Forms task)
- *Modify a topological point* (on page 1334) (in the Molded Forms task)
- *Change a topological point to a different point definition* (on page 1335) (in the Molded Forms task)
- *Delete a topological point* (on page 1335) (in the Molded Forms task)

Place a topological point on a surface and along a curve

1. In the Molded Forms task, select **Insert** > **Topological Points**.
2. Click **PointAlongCurveOnSurface** on the **Point Palette**.
3. Select a coordinate system.
4. Select a surface, such as a plate system or reference plane.
5. Select a plane intersecting the surface. The **Offset** direction is along the reference curve created by this intersection.
6. Select another plane intersecting the surface and the reference curve. The **Distance** direction is along the supporting curve created at the intersection with the surface.

7. Select a value for **Distance Method**.
8. Type the needed values for **Offset** and **Distance**.

   A preview point in green appears at the intersection of the surface, reference curve, and supporting curve. If more than one point appears, then more than one solution is possible.
9. If multiple points appear, perform one of the following steps to resolve the ambiguity:
   - If the surface is complex (a plane intersects it more than once), then select a coordinate axis direction for **Inter Ambiguity**.
   - If **Girth** is selected for **Distance Method**, then select a coordinate axis direction for **Point Ambiguity**.
   - If two solutions are possible on opposing sides of the reference curve, then select a coordinate axis direction for **Side Ambiguity**.

   NOTE For each ambiguity, select a value that is compatible with the orientation of the points.

   A single green preview point is now visible.
10. Optionally, type a name for the point.
11. Optionally, select **More** in the **Parent** box, and select a parent system for the point. For example, you can select the plate system on which you are placing the point.
12. Click **Finish**.

    *Point geometry is displayed in the graphic view and in the **Workspace Explorer**.*



Modify a topological point

1. On the vertical toolbar, click **Select** .
2. In a graphic view or in the Workspace Explorer, select the topological point to modify.

   The point is highlighted in the graphic view, the ribbon for the topological point appears, and inputs are highlighted in the Workspace Explorer.
3. Make the needed changes.
4. Click **Accept** .

   A preview of the changed geometric construction appears.
5. Click **Finish**.

   The modified point is displayed in the graphic view and in the Workspace Explorer.

Change a topological point to a different point definition

1. On the vertical toolbar, click **Select** .
2. In a graphic view or in the Workspace Explorer, select the topological point to modify.

   *The point is highlighted in the graphic view, the ribbon for the topological point appears, and the inputs are highlighted in the **Workspace Explorer**.*
3. In the **Macro Definition** box, click **More**.

   *The **Select Geometric Construction** dialog box appears. Only the appropriate point geometric constructions appear.*
4. Select the needed point geometric construction, and then click **OK**.

   *The new selection appears in the **Macro Definition** box with a yellow background. Options with blue backgrounds contain new values suggested by the software. Options with white backgrounds retain the values from the previous geometric construction.*
5. Make the needed changes to options on the ribbon.

   Changed options have a yellow background.
6. Click **Accept** .

   A preview of the changed point appears.
7. Click **Finish**.

   *The modified point is displayed in the graphic view and in the **Workspace Explorer**.*

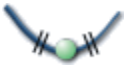
Delete a topological point

1. On the vertical toolbar, click **Select** .
2. In a graphic view or in the Workspace Explorer, select the topological point to delete.

   The point is highlighted in the graphic view, the ribbon for the topological point appears, and inputs are highlighted in the Workspace Explorer.
3. Click **Delete** .

   The geometric construction is deleted.

Point at Curve Middle



Tooltip: PointAtCurveMiddle

Creates a point at the middle of a curve.

Required Input	Output
Curve, line, or edge	Point
Optional Input	
Projection surface	

Point at Curve Middle Ribbon (for geometric constructions)

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the curve on which the point is placed.

(2)

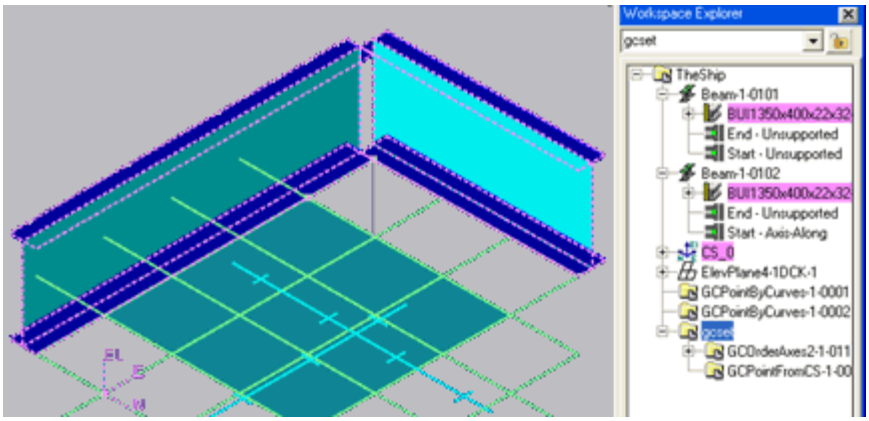
Specifies a surface on which to project the point. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point at Curve Middle Ribbon (for topological points in the Molded Forms task)

Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.**1** - Specifies the curve on which the point is placed.

(2)

Specifies a surface on which to project the point. This value is optional.

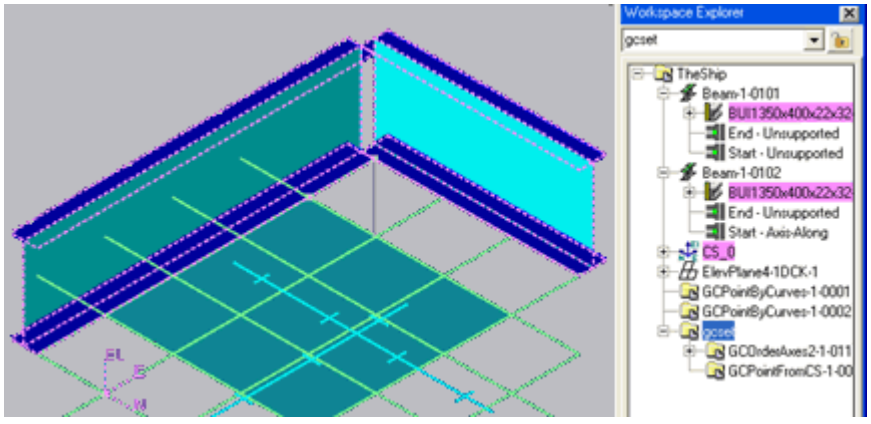
Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background when no input is selected.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

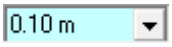
NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- 1 and 0.00 m - No background when the value of the original geometric construction is retained.
- 1 and 0.10 m - Yellow background when you change a value.

-  and  - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

-  - Blue background when the input is the same for all selected geometric constructions.
-  - No background with pink underline when the input is different for *all* selected geometric constructions.
-  - Blue background with pink underline when the input is different for *some* selected geometric constructions.
-  - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.
-  - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.



Reject

Clears the current selections.

Accept

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.
- **More** - Select a system in the model.

NOTE The **New GC Set** option is not used for topological points.

What do you want to do?

- *Place a point at the middle of a curve* (on page 1340)

- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)
- Place a topological point at the middle of a curve (in the Molded Forms task)
- *Modify a topological point* (on page 1334) (in the Molded Forms task)
- *Change a topological point to a different point definition* (on page 1335) (in the Molded Forms task)
- *Delete a topological point* (on page 1335) (in the Molded Forms task)

Place a point at the middle of a curve

1. Click **PointAtCurveMiddle** on the **Geometric Construction Palette**.
2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

   A preview of the point appears at the middle of the curve.
3. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointAtCurveMiddle** in the **Geometric Construction Explorer**.*
4. If needed, create additional geometric constructions.
5. Click **Close** to return to the main command ribbon.

IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.
- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point at Minimum Distance



Tooltip: PointAtMinimumDistance

Creates a point at defined distances from the faces of two plate, profile, or member systems.

Required Input	**Output**
Coordinate system	Point controlled by driven dimensions
Intersecting plate systems or surfaces	
Sketching plane	
Parameters	
Measurement method and distances	

Point at Minimum Distance Ribbon

Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

Specifies the coordinate system used for the specified directions and axes.

2

Specifies the first plate system, stiffener system, member system, or surface.

3

Specifies the second plate system, stiffener system, member system, or surface.

4

Specifies the sketching plane that intersects the selected systems or surfaces.

(5)

Specifies an auxiliary reference for the sketching plane, such as a plate or profile system.

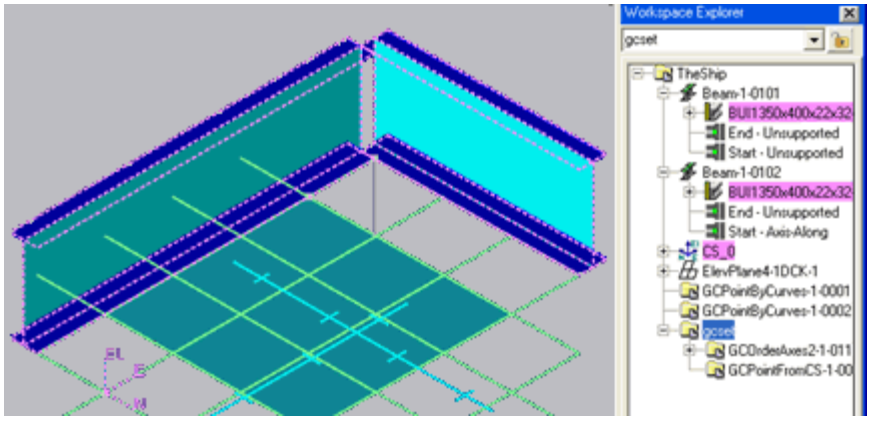
Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background when no input is selected.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- 1 and 0.00 m - No background when the value of the original geometric construction is retained.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

- 1 - Blue background when the input is the same for all selected geometric constructions.
- 1 - No background with pink underline when the input is different for *all* selected geometric constructions.

-  - Blue background with pink underline when the input is different for *some* selected geometric constructions.
-  - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.
-  - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.



 **Reject**

Clears the current selections.

 **Accept**

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

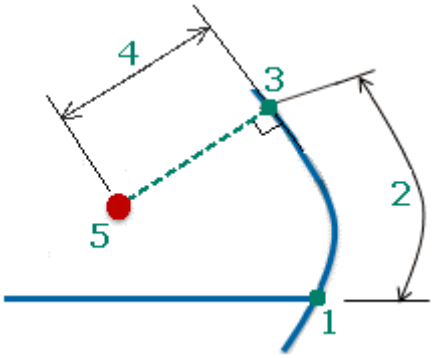
- The current model. This is the default value.
- **More** - Select a system in the model.

Method

Specifies the measurement method. Select **At girth then along normal** or **At intersection of offset curves**. Descriptions of the parameters for each method are below.

At girth then along normal Method

Positions a point at the girth distance from the starting point along the curve of the first system (such as the hull), and then at a distance from the girth point along the normal to the curve. The intersection point is at the intersection of the first system, the second system, and the sketching plane. The curve of the first system is at the intersection between the first system and the sketching plane.



1 - Intersection point

2 - Girth distance

3 - Girth point

4 - Normal distance

5 - Point at minimum distance

Girth Distance

Defines the *girth* (on page 1457) distance along the first system curve.

Normal Distance

Defines the linear distance normal to the first system curve.

Intersection

Defines the axis direction for the intersection point, when the first system, second system, and sketching plane intersect in more than one location. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. For example, a hull, deck, and transverse sketching plane intersect on the port and starboard sides. Select **+y** to use the starboard side starting point or **-y** to use the port side starting point.

Girth

Defines the axis direction used to orient the girth distance measurement.

Normal

Defines the axis direction used to orient the normal distance measurement.

Offset 0

Defines the distance that the point is offset from the sketching plane **4**.

Face Reference

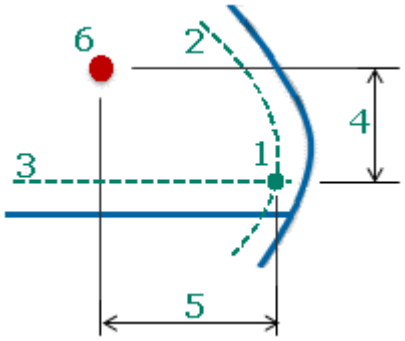
Specifies the profile face to use to create a sketching plane. Select **Web** or **Flange**. This option is only available when a profile system is selected for sketching plane **4**.

Direction

Specifies the direction of **Offset 0**, using the axis normal to the sketching plane. Select **Positive** or **Negative**.

At intersection of offset curves Method

Positions a point linearly along the select coordinate system axes from the intersection point of the offset curves from the first and second systems.



1 - Intersection point

2 - Offset 1 curve

3 - Offset 2 curve

4 - Offset 1 distance

5 - Offset 2 distance

6 - Point at minimum distance

Offset 1 Distance

Defines the linear distance along the first system curve offset.

Offset 2 Distance

Defines the linear distance along the second system curve offset.

Intersection

Defines the axis direction for the intersection point, when the first system, second system, and sketching plane intersect in more than one location. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. For example, a hull, deck, and transverse sketching plane intersect on the port and starboard sides. Select **+y** to use the port side starting point or **-y** to use the starboard side starting point.

Offset 1

Defines the axis direction used to orient the first offset measurement.

Offset 2

Defines the axis direction used to orient the second offset measurement.

Offset 0

Defines the distance that the point is offset from the sketching plane **4**.

Face Reference

Specifies the profile face to use to create a sketching plane. Select **Web** or **Flange**. This option is only available when a profile system is selected for sketching plane **4**.

Direction

Specifies the direction of **Offset 0**, using the axis normal to the sketching plane. Select **Positive** or **Negative**.

What do you want to do?

- *Place a topological point along a girth distance and offset normal to the curve* (on page 1346)
- *Place a topological point offset from the intersection of two offset curves* (on page 1347)
- *Modify a topological point* (on page 1334)
- *Change a topological point to a different point definition* (on page 1335)
- *Delete a topological point* (on page 1335)

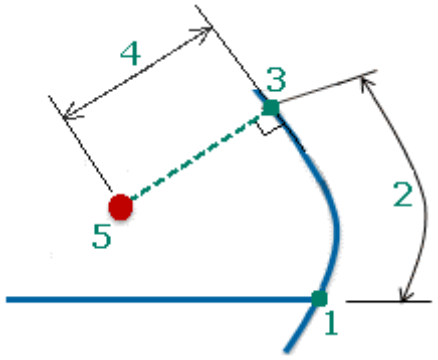
Place a topological point along a girth distance and offset normal to the curve

1. In the Molded Forms task, select **Insert** > **Topological Points**.
2. Click **PointAtMinimumDistance** on the **Point Palette**.
3. Select a coordinate system.
4. Select a plate system, profile system. member system, or surface. The girth distance is measured along this system.
5. Select a second plate system, profile system. member system, or surface. This selection must intersect or be bounded to the first system.
6. Select a sketching plane, such as a reference, that intersects the two systems.
7. In the **Method** box, select **At girth then along normal**.

   A point appears on the sketching plane at the intersection of the selected systems or surfaces. This is the intersection point.
8. Type a value for **Girth**.

   A point appears on the sketching plane along the girth distance of the first system. This is the girth point.
9. Type a value for **Normal**.

   The point moves from the girth at a distance normal to the curve of the system.



1 *- Intersection point*
2 *- Girth distance*
3 *- Girth point*
4 *- Normal distance*
5 *- Point at minimum distance*

10. Click **Finish**.

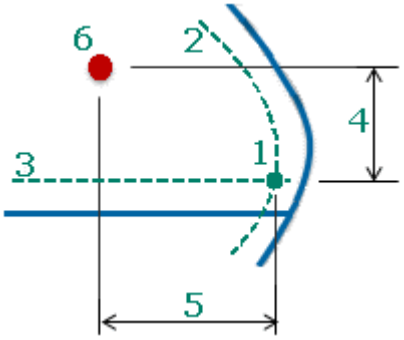
    Point geometry is displayed in the graphic view and in the Workspace Explorer.

Place a topological point offset from the intersection of two offset curves

1. In the Molded Forms task, select **Insert** > **Topological Points**.
2. Click **PointAtMinimumDistance** on the **Point Palette**.
3. Select a coordinate system.
4. Select a plate system, profile system. member system, or surface. The girth distance is measured along this system.
5. Select a second plate system, profile system. member system, or surface. This selection must intersect or be bounded to the first system.
6. Select a sketching plane, such as a reference, that intersects the two systems.
7. In the **Method** box, select **At intersection of offset curves**.

   A point appears on the sketching plane at the intersection of the selected systems or surfaces. This is the intersection point.
8. Type a value for **Offset 1 Distance**.
9. Type a value for **Offset 2 Distance**.

   The point moves the location of the offsets.

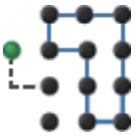


1 *- Intersection point*
2 *- Offset 1 curve*
3 *- Offset 2 curve*
4 *- Offset 1 distance*
5 *- Offset 2 distance*
6 *- Point at minimum distance*

10. Click **Finish**.

    Point geometry is displayed in the graphic view and in the Workspace Explorer.

Point at Offset from Key Point



Tooltip: PointAtOffsetFromKeyPoint

Creates a point using a profile or member cross-section key point as reference. For more information, see *Key Points* (on page 1353).

Required Input	**Output**
Profile stiffener system or member system cross-section	Point controlled by driven dimensions
Sketching plane	
Parameters	
Key point	
Offset distances	

Point at Offset from Key Point Ribbon

Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

Specifies the profile or member system.

2

Specifies a plate system, profile system, or reference plane used as a sketching plane intersecting profile or member system **1**.

(3)

Specifies a coordinate system. If no coordinate system is selected, then the main coordinate system is the default. This value is optional.

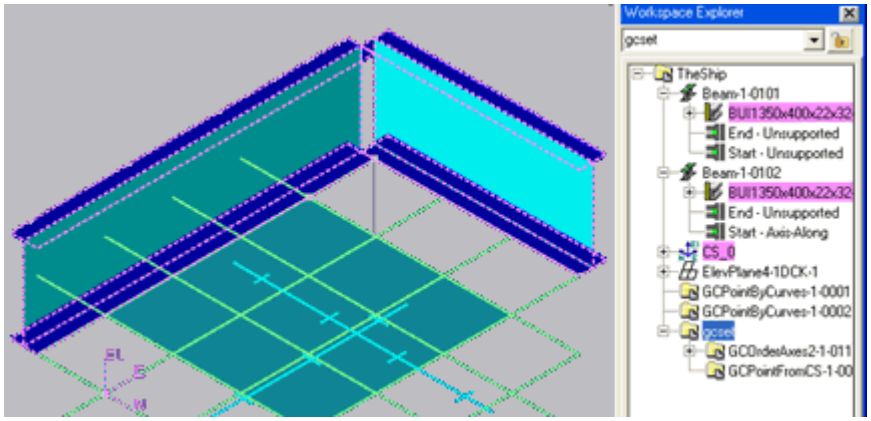
Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

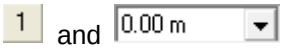
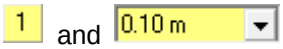
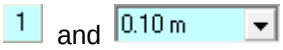
-  - No background when no input is selected.
-  - Yellow background when an input is selected.
-  - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

-  - No background when the value of the original geometric construction is retained.
-  - Yellow background when you change a value.
-  - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

-  - Blue background when the input is the same for all selected geometric constructions.
-  - No background with pink underline when the input is different for *all* selected geometric constructions.
-  - Blue background with pink underline when the input is different for *some* selected geometric constructions.

-  - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.
-  - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.



Reject

Clears the current selection.

Accept

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.
- **More** - Select a system in the model.

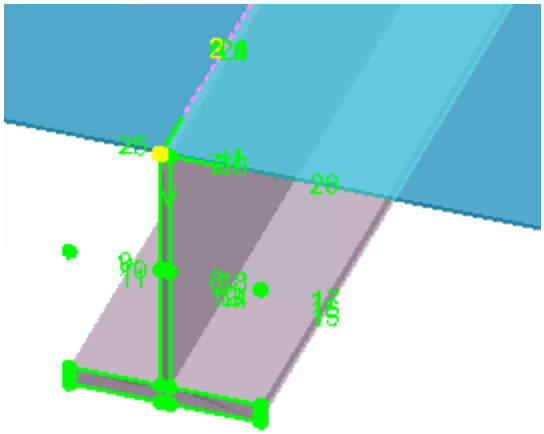
NOTE The **New GC Set** option is not used for topological points.

KeyPoint

Specifies a key point by index number. Every cross-section has 27 key points, although some are combined for cross-sections with simple geometry. For more information, see *Key Points* (on page 1353).

Select Value in the 3D Graphic View

Allows key point selection in the graphic view. Key points display as preview geometry, key point numbers, and the local u-v coordinate system. For more information, see *Key Points* (on page 1353).



U Offset

Specifies the offset of the geometric construction point from the key point along the local u-axis of the cross-section. For more information, see *Key Points* (on page 1353).

V Offset

Specifies the offset of the geometric construction point from the key point along the local v-axis of the cross-section. For more information, see *Key Points* (on page 1353).

Offset 0

Defines the distance that the point is offset from the sketching plane **4**.

Face Reference

Specifies the profile face to use to create a sketching plane. Select **Web** or **Flange**. This option is only available when a profile system is selected for sketching plane 2.

Direction

Specifies the direction of **Offset 0**, using the axis normal to the sketching plane. Select **Positive** or **Negative**.

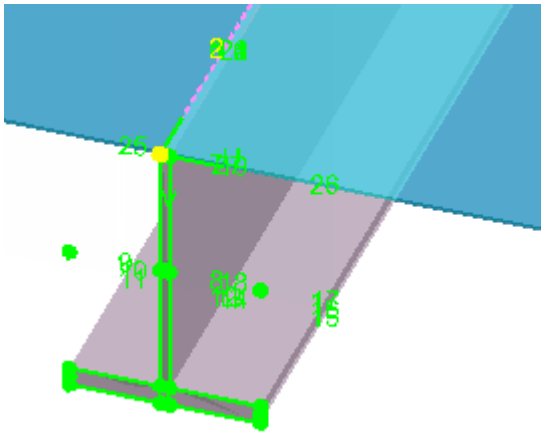
What do you want to do?

- *Place a topological point referenced from a profile or member cross-section key point* (on page 1352)
- *Modify a topological point* (on page 1334)
- *Change a topological point to a different point definition* (on page 1335)
- *Delete a topological point* (on page 1335)

Place a topological point referenced from a profile or member cross-section key point

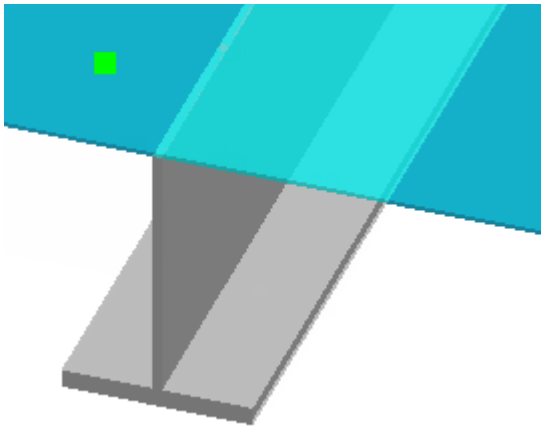
1. In the Molded Forms task, select **Insert** > **Topological Points**.
2. Click **PointAtOffsetFromKeyPoint** on the **Point Palette**.
3. Select a profile or member system.
4. Select a plate system, profile system, member system, or reference plane intersecting the first profile or member system. This is the sketching plane.
5. Select an index number in the **Keypoint** box, or click **Select Value in the 3D Graphic View** and select a key point in the graphic view.

   The key point displays in yellow.



6. Type u-axis and v-axis offset values.

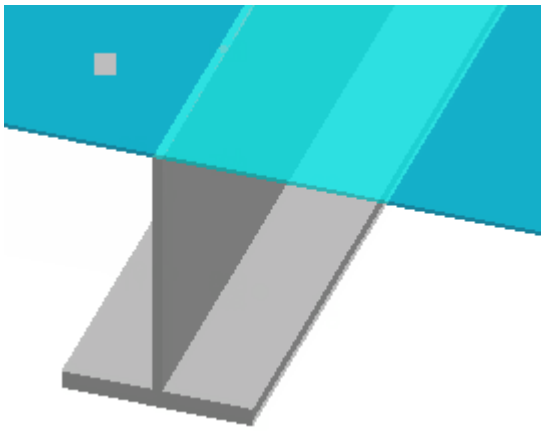
   Preview geometry of the point displays in green.



7. Optionally, type a name for the point.
8. Optionally, select **More** in the **Parent** box, and select a parent system for the point. For example, you can select the profile or member system.

9. Optionally, type a value for the offset from the sketching plane in **Offset 0**, and select a value for **Direction**.
10. Click **Finish**.

    Point geometry is displayed in the graphic view and in the Workspace Explorer.



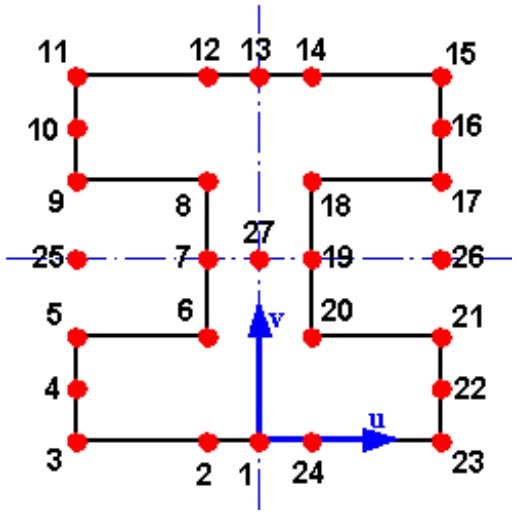
Key Points

NOTE Profile cross-sections use a numbering system called key points. Members use an equivalent point system called cardinal points. The numbering convention for cardinal points differs from key points, as described below.

Key Point Numbering Conventions

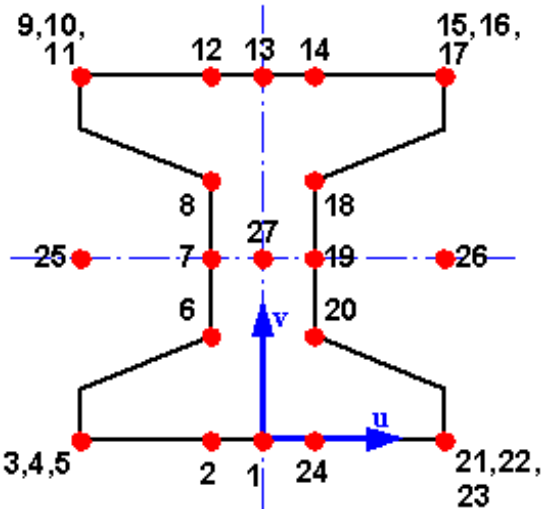
Key points are numbered consistently among all types of profile cross-sections without regard to the number of geometric elements. This assures consistent behavior in the software when changing cross-sections of an existing profile system. As a result, some points are coincident on some cross-sections. Key point numbering is based on an I-beam profile because it is the most complex shape. Numbering for other profiles is derived from the I-beam.

I-beam

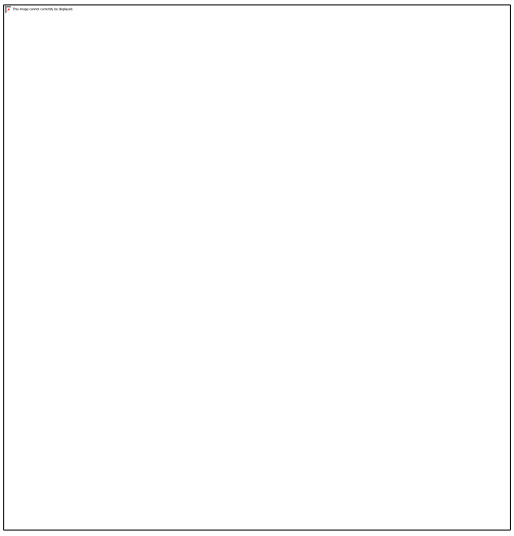


Includes rolled and built-up cross-sections.

S-beam

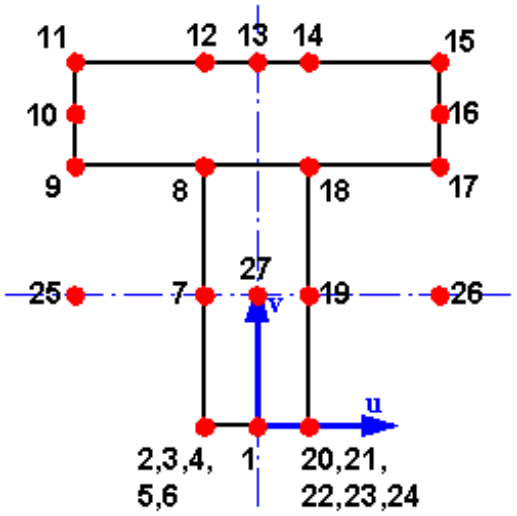


T



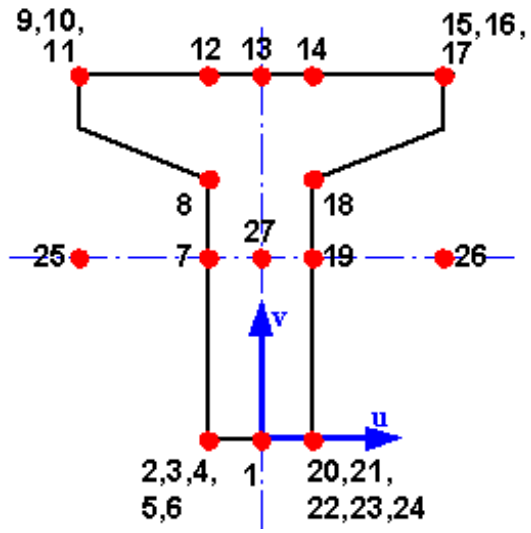
Includes T and I-to-T. Midpoint is from top of flange.

Fabricated T



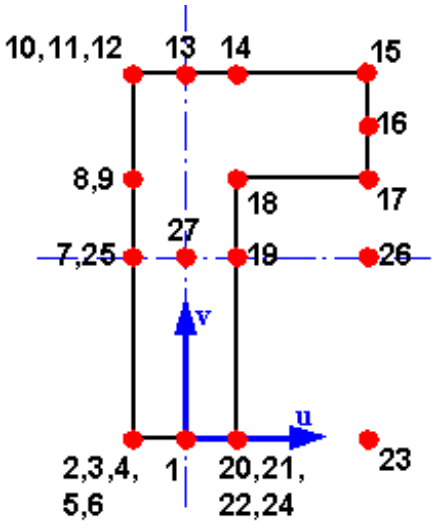
Midpoint is from bottom of flange.

S-T



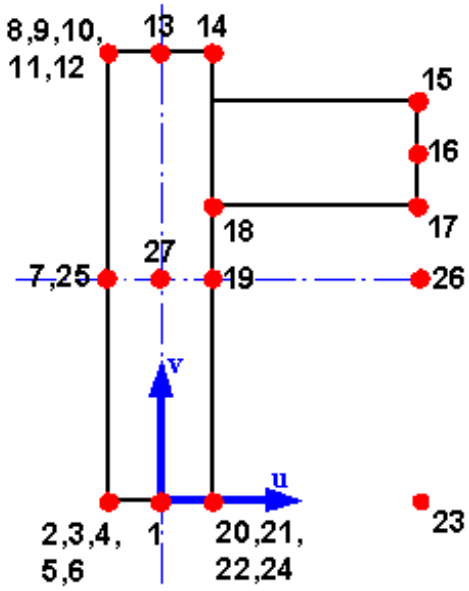
Midpoint is from top of flange.

Angle

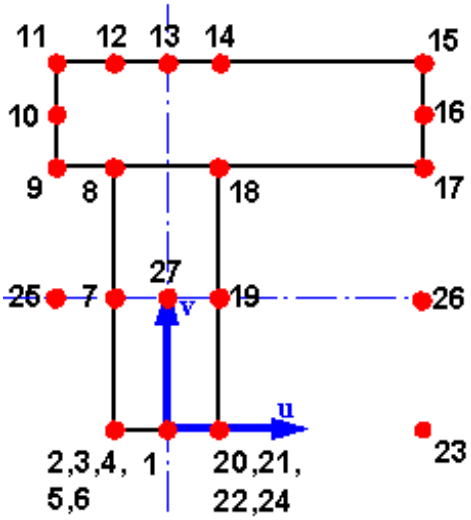


Midpoint is from top of flange.

Fabricated angles

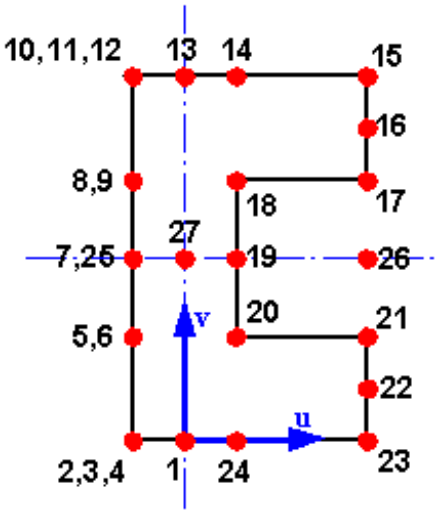


Midpoint is from top of flange.

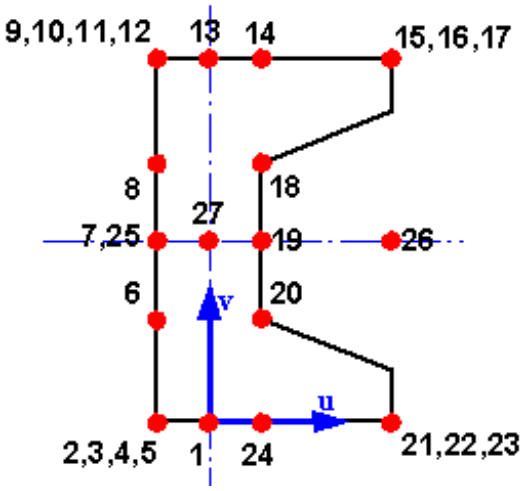


Midpoint is from bottom of flange.

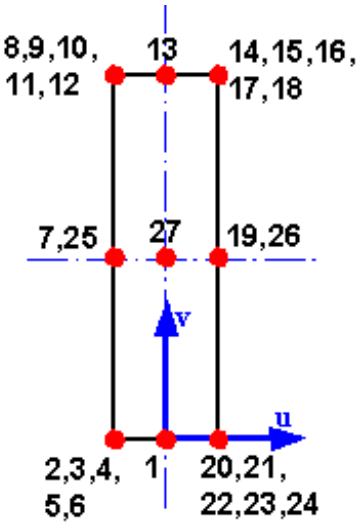
Channel



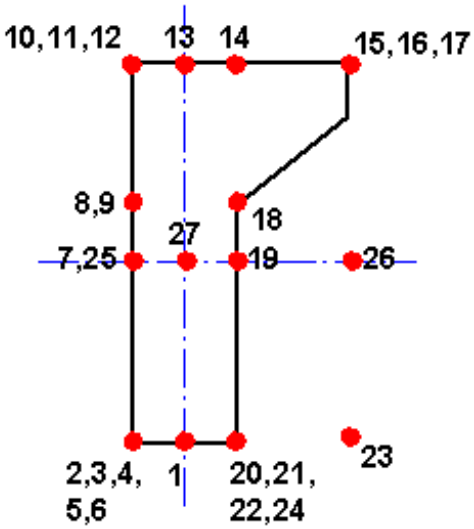
S-channel



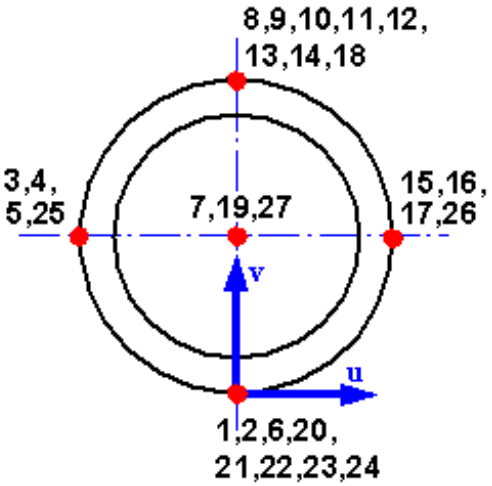
Flat bar



Bulb

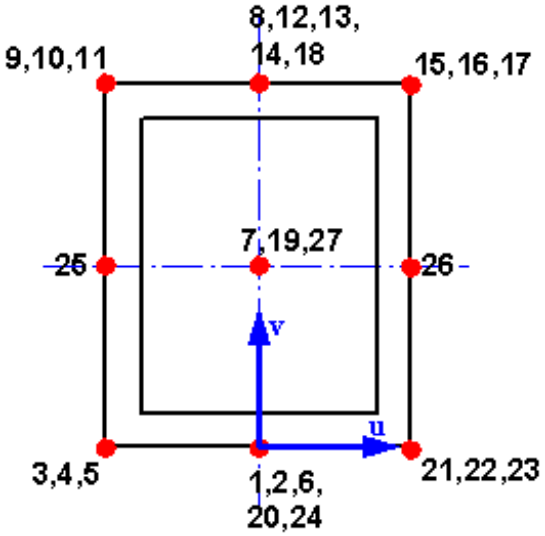


Pipe

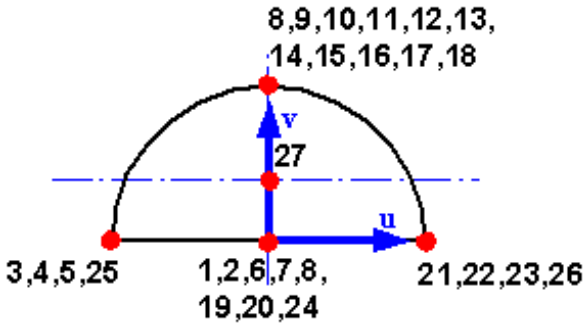


Includes pipe, tube, and round bar.

Rectangular tube



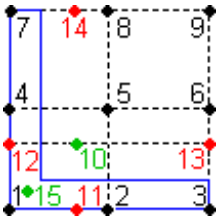
Half-round



Cardinal Point Numbering Conventions

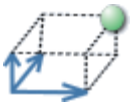
A cardinal point defines the location where the software displays a member cross-section. There are 15 cardinal points definable for a member cross-section symbol, but only one can be used at a time when the member is placed. Each cardinal point has a number and a name.

The local y-axis of the member and the center-of-gravity point of the section define load points 11 and 14. The local z-axis of the member and the center-of-gravity point of the section define load points 12 and 13. The location of points 10 (center-of-gravity) and 15 (shear center) depend on the section shape.



1. Bottom Left
2. Bottom Center
3. Bottom Right
4. Center Left
5. Center
6. Center Right
7. Top Left
8. Top Center
9. Top Right
10. Centroid
11. Centroid Bottom
12. Centroid Left
13. Centroid Right
14. Centroid Top
15. Shear Center

Point from Coordinate System



Tooltip: PointFromCS

Creates a point at the defined X, Y, and Z distances from the origin of the selected coordinate system or from a selected point.

Required Input	**Output**
Coordinate system	Point
Optional Input	
Reference point	
Parameters	
X, Y, and Z distances	

Point from Coordinate System Ribbon (for geometric constructions)

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the coordinate system.

(2)

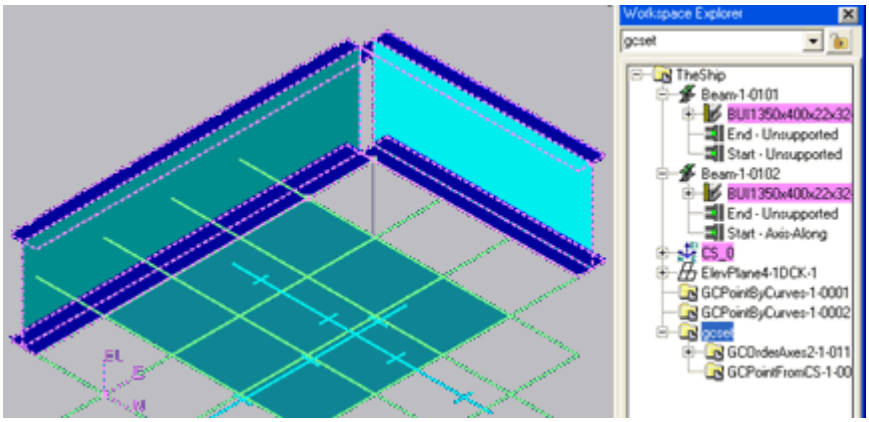
Specifies a reference point from which X, Y, and Z distances are measured. If no reference point is selected, the X, Y, and Z distances are measured from the origin of the coordinate system. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

X

Specifies the distance along the X-axis from the origin or reference point.

Y

Specifies the distance along the Y-axis from the origin or reference point.

Z

Specifies the distance along the Z-axis from the origin or reference point.

Point from Coordinate System Ribbon (for topological points in the Molded Forms task)

Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select a different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

Specifies the coordinate system.

(2)

Specifies a reference point from which X, Y, and Z distances are measured. If no reference point is selected, the X, Y, and Z distances are measured from the origin of the coordinate system. This value is optional.

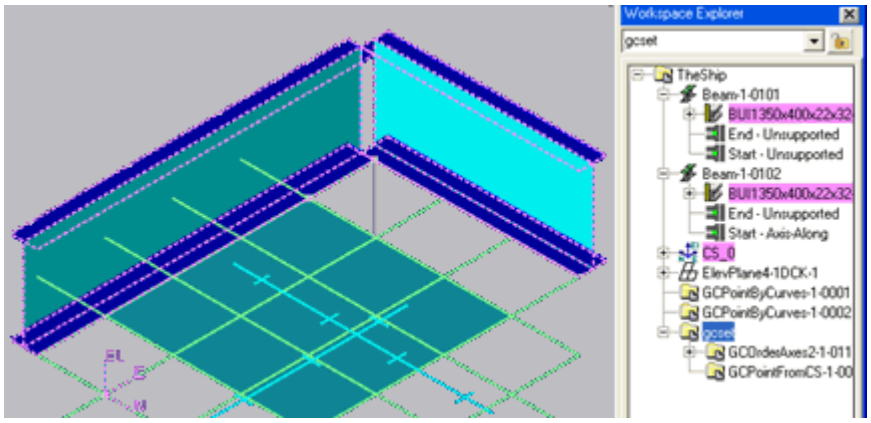
Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background when no input is selected.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- 1 and 0.00 m - No background when the value of the original geometric construction is retained.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

- 1 - Blue background when the input is the same for all selected geometric constructions.
- 1 - No background with pink underline when the input is different for *all* selected geometric constructions.
- 1 - Blue background with pink underline when the input is different for *some* selected geometric constructions.
- 1 - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.
- - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.



Reject

Clears the current selections.

Accept

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.
- **More** - Select a system in the model.

NOTE The **New GC Set** option is not used for topological points.

X

Specifies the distance along the X-axis from the origin or reference point.

Y

Specifies the distance along the Y-axis from the origin or reference point.

Z

Specifies the distance along the Z-axis from the origin or reference point.

What do you want to do?

- *Place a point by distances from a coordinate system origin* (on page 1045)
- *Place a point by distances from a reference point* (on page 1045)
- *Modify a geometric construction* (on page 1055)
- *Change a geometric construction to a different definition* (on page 1056)
- *Delete a geometric construction* (on page 1057)
- *Place a topological point by distances from a coordinate system origin* (on page 1364) (in the Molded Forms task)
- *Place a topological point by distances from a reference point* (on page 1364) (in the Molded Forms task)
- *Modify a topological point* (on page 1334) (in the Molded Forms task)
- *Change a topological point to a different point definition* (on page 1335) (in the Molded Forms task)
- *Delete a topological point* (on page 1335) (in the Molded Forms task)

Place a topological point by distances from a coordinate system origin

1. In the Molded Forms task, select **Insert** > **Topological Points**.
2. Click **PointFromCS** on the **Point Palette**.
3. Select a coordinate system.
4. Type values for **X**, **Y**. and **Z**.

   *A green preview point appears. It is offset from the coordinate system origin by the **X**, **Y**. and **Z** values.*
5. Optionally, type a name for the point.
6. Optionally, select **More** in the **Parent** box, and select a parent system for the point.
7. Click **Finish**.

   Point geometry is displayed in the graphic view and in the Workspace Explorer.



Place a topological point by distances from a reference point

1. Create a point to use as a reference point by creating another topological point or using **Insert** > **Control Point**.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) (on page 1323) in the *Molded Forms User's Guide*
2. In the Molded Forms task, select **Insert** > **Topological Points**.
3. Click **PointFromCS** on the **Point Palette**.
4. Select a coordinate system.
5. Click (2), and select the reference point.
6. Type values for **X**, **Y**, and **Z**.

   *A green preview point appears. It is offset from the reference point by the **X**, **Y**, and **Z** values and along the **X**, **Y**, and **Z** axes of the coordinate system.*
7. Optionally, type a name for the point.
8. Optionally, select **More** in the **Parent** box, and select a parent system for the point.
9. Click **Finish**.

Point geometry is displayed in the graphic view and in the Workspace Explorer.



SECTION 29

Delay Settings (Tools Menu)

Tools > Delay Settings specifies whether changes made in Molded Forms are immediately updated in Structural Detailing or Structural Manufacturing. When the delay is active, changes to the Molded Form systems appear in Molded Forms, but changes needed to the corresponding Structural Detailing and Structural Manufacturing objects are put on the **To Do List.** When the delay is not active, the changes needed to the corresponding Structural Detailing and Structural Manufacturing objects are propagated immediately.

NOTE This setting is saved in, and applies only to, your session file. Therefore, it is possible that objects that you expect to appear in the **To Do List** propagate instead because they are used in someone else's session where the delay setting is off.

This command opens the **Delay Operations** dialog box.

What do you want to do?

- *Turn on delay operations* (on page 1366)
- *Update delayed objects* (on page 1366)
- *Turn off delay operations* (on page 1366)

Turn on delay operations

1. Click **Tools > Delay Settings**.
2. Click the **Delay Settings** tab.
3. Turn on the **Delay Structure Operations** options that meet your requirements.
4. Click **OK**.

Update delayed objects

1. Click **Tools** > **Delay Settings**.
2. Click the **Update To Do Records** tab.
3. Define the record filter as required.
4. Type a file name and folder path for the error file.
5. Click **Update Objects**.

Turn off delay operations

1. Click **Tools** > **Delay Settings**.
2. On the **Delay Settings** tab, click **No Delay**.

3. Click **OK**.

Delay Operations Dialog Box

Displays settings for the **Delay Settings** command.

Delay Settings Tab (Delay Operations Dialog Box) (on page 1367)
Update To Do Records Tab (Delay Operations Dialog Box) (on page 1367)
To Do Delay Record Detail Tab (Delay Operations Dialog Box) (on page 1368)

Delay Settings Tab (Delay Operations Dialog Box)

Specifies the delay operation, if any, that you want taken.

Report Data Generation

Disable Report Data

Select to disable the report data generation.

Delay Structure Operations

No Delay

Select this option to have changes to the Structural Manufacturing and Structural Detailing objects propagated when the Molded Form object changes.

Delay Molded Forms

Select this option to have changes to the Molded objects sent to the **To Do List** instead of being propagated.

Delay Struct Detailing

Select this option to have changes to the Structural Detailing objects sent to the **To Do List** when the Molded Form object changes.

Update To Do Records Tab (Delay Operations Dialog Box)

Controls how objects that you have delayed to the **To Do List** are updated.

To Do Record Filters

User Name

Select the user name to filter the **To Do List** records by.

Dates

Define the starting (**From**) and ending (**To**) dates to filter the **To Do List** records by.

Order

Select the order in which to process the **To Do List** records. Select **None**, **Optimal**, **Ascending**, or **Descending**.

TIP Select **Optimal** to allow the software to pick the order of updating that most logically clears records from the **To Do List**.

Type

Select which type of **To Do List** records to process. Select **Errors**, **Warnings**, **Out-of-Date**, **Out-of-Date Reporting**, or **Out-of-Date Molded Forms**.

Structure to Update

Select **All**, **Hull**, or **Select a Filter.**

- **All** or **Hull** updates the objects accordingly.
- **Select a Filter** launches additional options, **CreateNewFilter** and **More(Select from Existing Filters)**. Both options require you to select a filter before the system can update the objects.

Update To Do Records

Update Objects

Click to update all **To Do List** records that satisfy the filter you have defined. Objects are updated in the selected order.

Cancel Update

Stops the update process.

Batch Update

Click to update all **To Do List** records that meet the requirements for the filters that you have defined through the **Batch** process. The system updates objects in the optimal order.

Log

Log File

Type a file name and folder path for the error log.

Enable Logging

Turn on to generate the log file.

View Log File

Click to display the error log.

To Do Delay Record Detail Tab (Delay Operations Dialog Box)

Displays detailed information about the delayed **To Do List** records.

SECTION 30

Repairing Tribon Data

The Tribon translator software includes a command that you can use to repair the data in Tribon XML files. Examples of data to repair include:

- Instance properties
- Profile part and seam location

★IMPORTANT The integrity of the source data is the responsibility of the generating software (in this case, Tribon). When you repair the data with this command, you assume liability and responsibility for the results. Intergraph does not assume any liability for the data modifications.

See Also

Repair Tribon XML (Tools Menu) (on page 1369)

Repair Tribon XML (Tools Menu)

Tools > **Repair Tribon XML** modifies data in Tribon XML files. You can then import the modified data into Smart 3D.

Repair Tribon XML Dialog Box **(on page 1369)**

Repair Tribon data

1. Import Tribon data from an XML file. For more information, see *Import Ship Structure* (on page 1231).
2. If you see errors in the imported geometry, perform the following steps.
3. Click **Tools** > **Repair Tribon XML**.
4. On the **Repair Tribon XML** dialog box, specify the input file, output file, and log file for the repair process.
5. Select the curved panels to repair.
6. Check the repair options as needed.
7. Click **OK**.
8. Re-import the data using the repaired XML file. Objects that are different from the previous import are automatically updated in the model.

Repair Tribon XML Dialog Box

Sets options for repairing Tribon data.

Files

Input XML file

Specify the import file.

Output XML file

Specify the output file. You can import this file into Smart 3D using the **File** > **Import** > **Ship Structure** command.

Log File

Specify a log file that stores information about the repair process.

Panel

Curved Panels

Select this box to choose curved panels, which are panels defined on a sculpted, not flat, surface.

Plane Panels

This option is not available in this release.

Select the curved panels to be modified in the xml

Click the panels to repair.

Options

Change instance property of panel to As Defined

Changes the instance property to **AsDefined**. This property specifies the position of objects relative to their defining geometry. The different values for this property can be:

- **AsDefined** - The objects are located where they were defined.
- **Reflected** - If the objects were defined on the port side, they are actually on the starboard side. If the objects were defined on the starboard side, they are actually on the port side.
- **Both** - The objects are symmetrical with respect to the origin and are located on both sides.

Move profile parts to other side of center line (Change Y value)

Reflects the profile parts about the Y-axis so the parts are in the correct location.

Move seams to other side of centerline (Change Y value)

Reflects the seams about the Y-axis so the parts are in the correct location.

The other options in this section are not available in this release of the software.

SECTION 31

Data Integrity Check (Tools Menu)

The **Tools** > **Data Integrity Check** command checks the attributes, ports, boundaries, seams, thickness direction, and connections of plates to verify their integrity. Although every precaution has been taken in the software to ensure the integrity of all plates, plate corruption is possible. This command allows you to search for any corruption or errors.

You can view errors interactively by using the *Results Dialog Box* (on page 1372).

Data Integrity Check Ribbon

Displays the options that you use to select objects in the model to check, and to view the results of the check.



Properties

Opens the **Properties** dialog box, which you use select the name and location of the log file created by the **Data Integrity Check** command.



Select Objects

Select the objects to check. Selection is controlled by the **Part Selection** option.

Finish

Performs the data integrity check on selected objects.

Part Selection

Select the selection method from the following options:

- **Workspace** - Selects all objects in the workspace.
- **Select Parts** - Manually select parts in the **Workspace Explorer** or a graphic view.
- **Database** - Selects all objects in the model database.
- **Last Results** - Opens the **Results** dialog box. After making a part selection and clicking **Finish** in the current session, the results of the data integrity check are displayed. For more information, see *Results Dialog Box* (on page 1372).

Check data integrity of plates

1. Click **Tools** > **Data Integrity Check**.
2. Click **Properties**  .
3. Define a file name and folder location for the log file.
4. Click **OK**.
5. In the **Part Selection** list, select the method used to find plates to check.
6. Click **Finish**.
7. After processing is complete, click **View** in the **View Log** dialog box.

*The **Results** dialog box opens. Each object with an error message is a row on the grid.*

TIP The **Results** dialog box can be reopened at any time in the current session by selecting **Last Results i**n the **Part Selection** option.

8. Select **Auto Zoom.**
9. Select a row in the grid.

   A new graphic view window opens. The new graphic view zooms to and highlights the affected objects. The objects affected by the error also highlight in existing graphic views without zooming.

10. Right-click the row and select **Help**.

    ***Molded Forms Help** opens to a page describing the error, possible causes, and solutions.*

    TIP You can also press **F1,** or click **Help** in the **Selection options** frame.

11. To correct the error, follow the instructions in **Molded Forms Help**.

Results Dialog Box

The **Results** dialog box displays error messages created from carrying out the **Data Integrity Check** command. It also provides tools for filtering the results and correcting the errors.

You open this dialog box by clicking **Last Results** in the **Part Selection** list on the Data Integrity Check ribbon.

NOTE Results are available only with the current session. If you start a new session, click **Tools** > **Data Integrity Check** again to see results in the **Results** dialog box.

Filter Options

The results can be filter using the following options:

Filter selection

Select one or more object type filters.

Enable filters

Select to allow selection of filters in the **Filter selection** list.

Select All

Click to select all filters in the **Filter selection** list.

Select None

Click to clear selected filters in the **Filter selection** list.

Apply

Click to apply selected filters to the results in the grid. Only results meeting the filter criteria appear in the grid.

Selection Options

A result selected in the grid can be viewed or repaired using the following options:

Edit

Selects the object named in the selected row, in both the graphic view and the **Workspace Explorer**. You can now edit the object to correct the error. This command is also available on the shortcut menu.

Recompute

Recalculates the object and all related objects. Performs the same recalculation as the **Tools** > **Recompute Objects** command. For more information, see Recompute Objects. This command is also available on the shortcut menu.

Help

Opens **Molded Forms Help** to the page describing the error, possible causes, and solutions for the object named in the selected row. This command is also available on the shortcut menu.

Properties

Opens the **Properties** dialog box for the object named in the selected row. This command is also available on the shortcut menu.

Auto Zoom

Select to open a second graphic view window. When you select a row, the new graphic view zooms to and highlights the affected objects. The objects affected by the error also highlight in existing graphic view without zooming.

Color 1

Sets the display color for the object named in the selected row.

Color 2

Sets the display color for shortcut menu commands.

Grid Properties

Lists the objects with errors. When you select a row, the object highlights in existing graphic views.

Name

Names of affected objects.

Status

The severity of the error. **Error** indicates a problem that must be fixed. **Warning** indicates an irregularity that may not be serious, but requires checking.

Comments

A description of the error.

Checked

The status of your actions on the affected object. To set the value to **Checked** or **Not checked**, select a row, right-click, and select **Toggle Checked/Unchecked**.

NOTE Click the property heading in the grid to sort by that property in ascending or descending order.

Shortcut Menu

Available by right-clicking a row.

Edit

Selects the object named in the selected row, in both the graphic view and the **Workspace Explorer**. You can now edit the object to correct the error.

Properties

Opens the **Properties** dialog box for the object named in the selected row.

Recompute

Recalculates the object and all related objects. Performs the same recalculation as the **Tools** > **Recompute Objects** command.

Update

Recalculates the object, more quickly but less thoroughly than **Recompute**. Performs the same recalculation as **Update**  on the **To Do List**. For more information, see To Do List Dialog Box.

Seam

Available for seam objects

- **View Bounded Seams** - Highlights seams bounded to the selected seam.
- **View Boundaries** - Highlights boundary objects for the selected seam.

Toggle Checked/Unchecked

Changes the value in the **Checked** box between **Checked** and **Not checked**.

Help

Opens **Molded Forms Help** to the page describing the error, possible causes, and solutions for the object named in the selected row.

See Also

Data Integrity Check (Tools Menu) (on page 1371)
Data Integrity Check Messages (on page 1407)

SECTION 32

Manufacturing Service Manager

Manufacturing Service Manager (in the Structural Manufacturing task) or **Tools** > **Manufacturing Service Manager** reviews and updates manufacturing objects. **Review** displays a report of the current condition of the manufacturing objects that you have selected. **Updating** pulls any outstanding changes into the manufacturing environment. The command is available on the vertical toolbar in the Structural Manufacturing task, and by clicking **Tools** > **Manufacturing Service Manager** in the Molded Forms, Structural Detailing, Planning, and Structural Manufacturing tasks.

You can select individual parts, assemblies, or blocks for this command.

The **Manufacturing Service Manager** pulls changes into the manufacturing environment instead of associatively pushing from other tasks into the manufacturing environment. In this way, the manufacturing environment is similar to the drawing environment; you have control over which objects get updated and when.

NOTES

- A manufacturing object goes on the **To Do List** for the following reasons:
  - The object has a nonworking status, or it contains objects which are included in different permission groups. You must change the manufacturing part status back to **Working**, and then update the manufacturing part by using the **Manufacturing Service Manager**.
  - The parent system of the manufacturing object was split in the Molded Forms task, and that split migration produced an error.
  - A compute found a failure within the rule and the manufacturing plate, manufacturing profile, or manufacturing member returned a warning.
  - Some other compute failure, with a specific action per entity, occurred.
- The same behavior occurs when the molded form split is performed by someone who does not have write permissions to the manufacturing part. In this case, a second person with write permission to the manufacturing part must update the manufacturing part by the **Manufacturing Service Manager**.

Manufacturing Service Manager Ribbon

Displays the controls used to update and review manufacturing objects.

Selection of Possible Report Data

Opens a dialog box from which you can set up reports.

Select

Selects objects in the model.

Finish

Starts the **Manufacturing Service Manager**.

Cancel

Clears the current selection.

Accept

Accepts the current selection.

Action

Determines whether the **Manufacturing Service Manager** updates the objects, reviews the objects, or reports the Detailed Review of the objects.

Filter

Specify the type of manufacturing part to use.

- **Parts** - All plate, profile, and member parts.
- **Parts or Assemblies** - All plate, profile, member, and different assembly types. For assemblies, you can select blocks, assembly blocks and assemblies.
- **Plate Parts** - Plate parts.
- **Plate Parts and Connected Profiles** - Includes all profiles connected to the plate parts.
- **Profile Parts** - Profile parts.
- **Member Parts** - Member parts.
- **Systems** - All plate systems, profile systems, and member systems.
- **Systems or Assemblies** - All systems and assemblies.
- **Plate Systems** - All plate systems.
- **Plate Systems and Connected Profiles** - All member systems.
- **Profile Systems** - All profile systems.
- **Member Systems** - All member systems and design member systems.

Connected

Specify which type of connected parts to use.

- **<Ignore>** - Only the selected part.
- **Direct Connected** - Includes objects that are physically connected to the selected part.
- **Indirect Connected** - Includes direct objects and objects that are physically connected to the direct objects.

What do you want to do?

- *Review a manufacturing object* (on page 1377)
- *Update a manufacturing object* (on page 1377)
- *Review or update a set of objects selected by a filter* (on page 1377)
- Detail Review a manufacturing object

Review a manufacturing object

1. Click **Select** on the vertical toolbar.
2. Select the manufacturing objects to review.

   TIP You can select the objects in the model or from the **Workspace Explorer**.
3. Click **Review** in the **Action** box.
4. Select the type of manufacturing object to review in the **Report Per** box.
5. Click **Finish**.

Update a manufacturing object

1. Click **Select** on the vertical toolbar.
2. Select the manufacturing objects to review.

   TIP You can select the objects in the model, or from the **Workspace Explorer**.
3. Click **Update** in the **Action** box.
4. Select the type of manufacturing object to review in the **Report Per** box.
5. Click **Finish**.

TIP The software updates the database as each manufacturing object is updated. Clicking **Cancel** stops the update for any objects that have not yet been posted to the database.

Review or update a set of objects selected by a filter

1. Click **More** from the filter selection list.

   *The **Select Filter** dialog box displays.*
2. Set up the filter to meet your requirements. For example, you could define a filter to select plate and profile parts in a specific block.

   The software finds and highlights the objects that match the filter.
3. Click **Service Manager** (in the Structural Manufacturing task) or **Tools** > **Manufacturing Service Manager**.

   The software prompts you to use the selected list of objects.
4. Click **Yes**.
5. If required, add additional objects to the list.
6. Click **Review** or **Update** in the **Action** box.
7. Select the type of manufacturing object to review in the **Report Per** box.
8. Click **Finish**.

Review Dialog Box

Displays the controls used to review the current manufacturing status of manufacturing objects.

NOTE This dialog box opens when you click **Finish** on the **Manufacturing Service Manager Ribbon**.

The manufacturing object status display includes:

- **Up to Date** - Indicates that the object does not require updating.
- **Out of Date** - Indicates that the object requires updating.
- **Can Be Deleted** - Indicates that the object is no longer needed.
- **To-Do-List (Error)** - Indicates that the object is on the **To Do List** because of an error condition.
- **To-Do-List (Out-of-Date)** - Indicates that the object is on the **To Do List** because it requires updating.
- **To-Do-List (Warning)** - Indicates that the object is on the **To Do List** because of a warning condition.
- **Edited** - Indicates that the object has been edited with the part editor. If you modify this part by using the **Manufacturing Plate** or **Manufacturing Profile** commands, or use the **BO Recompute** command, then the status does not display as **Edited**.



Properties

Displays the **Selection of Possible Report Data** dialog box. For more information, see *Selection of Possible Report Data Dialog Box* (on page 1379).

Report Per

Displays the current settings for the report data that is set to display. The information that displays depends on what you have set to show or hide on the **Selection of Possible Report Data** dialog box.

Update

Updates the objects. Only the objects shown on the dialog box are updated. This option does not update parts with a status of **Edited**.

Cancel

Closes the dialog box.

File name

Type the path and file name for the report file. You can use the **Browse** button beside the box to browse through the file system.

View

Displays the report file in the default text editor, such as **Notepad**.

Selection of Possible Report Data Dialog Box

Displays the controls used to determine which report data displays on the **Review** dialog box.

Attributes Tab

Option

Determines the manufacturing object type for which you are setting the review options. Select the box next to the option, and change the setting to **Show** or **Hide** as necessary.

Settings Tab

Review Settings

Turn on the settings to apply when the **Action** is set to **Review**. The available options include:

- Only include Out of Date manufacturing objects
- Only include manufacturing objects in **To Do List**
- Check detailed part in **To Do List**
- Include light parts
- Refresh after Update
- Highlight Out of Date objects

Update Settings

Turn on the settings to apply when the **Action** is set to **Update**. The available options include:

- Delete objects under light part
- Update objects with detailed part in **To Do List**
- Update manufacturing objects in **To Do List**
- Recompute all manufacturing objects

SECTION 33

Submit Batch Job

Tools > **Submit Batch Job** runs batch processes using the Intergraph Batch Services framework.

This command runs the following types of batch jobs:

Detailing:

- Detail parts
- Undetail parts
- Update reports

Production:

- Generate manufacturing parts
- Generate XML output files
- Manufacturing Service Manager update

Custom batch processes:

- BO recompute
- Other custom batch processes that you define.

What do you want to do?

- *Run a detailing batch process* (on page 1380)
- *Run a production batch process* (on page 1381)
- *Run a custom batch process* (on page 1381)
- *Add a custom batch process* (on page 1382)

Run a detailing batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Detailing** from the **Batch Process Type** list.

   *Smart 3D displays the available detailing batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

*Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.
6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Run a production batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Production** from the **Batch Process Type** list.

   *Smart 3D displays the available production batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.
6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Run a custom batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Custom Batch PRocesses** from the **Batch Process Type** list.

   *Smart 3D displays the available custom batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.
6. Click **Schedule**.

*The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Add a custom batch process

1. Add the ProgID and job description of the custom batch process to the <Product Folder>\CommonShip\SOM\Client\Xml\CustomBatchJobDetailsConfiguration.xml file.

   For example:

```
<JOB>
   <JobType>CustomBatchProcessD</JobType>
   <JobType_LocaleID>103</JobType_LocaleID>
   <JobDescription>Description of
CustomBatchProcessD</JobDescription>
   <JobDescription_LocaleID>104</JobDescription_LocaleID>
   <JobProgId> ProgID of CustomBatchProcess </JobProgId>
   <IsModelSpecific>False</IsModelSpecific>
</JOB>
```

2. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

3. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

4. Select **Custom Batch Process** from the **Batch Process Type** list.

   *The available custom batch processes display in the **Select Batch Process** list.*

5. Select the custom process to run, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

6. Specify the **Batch Input Type**, and add inputs as necessary.
7. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

8. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Generic Batch Command Dialog Box

Displays the controls used to submit a generic batch command.

Batch Process

Specifies the batch process to run. The list contains recently-selected job types. Click **More** to display the **Batch Process Selection** dialog box, and select a process that is not on the list. For more information, see *Batch Process Selection Dialog Box* (on page 1384).

Batch Input Type

Indicates the input type for the batch process.

- **Filter** indicates that you are providing a filter as an input to the batch job.
- **Assembly** indicates that you are providing an assembly as input to the batch job.
- **Object** indicates that you are providing a set of objects as input to the batch job.

 **Add**

Adds inputs to your batch job.

- If **Filter** is selected, this button displays the **Select Filter** dialog box so that you can add filters as inputs to your batch job.
- If **Assembly** is selected, this button displays the **Select Assembly** dialog box so that you can add assemblies as inputs to your batch job.
- If **Object** is selected, this button is not available. To add objects as inputs, select them from the model or the **Workspace Explorer**.

 **Remove**

Removes the selected inputs from the list. The software only removes the inputs that have the associated check box selected.

Share

Saves the inputs for the next batch process. If this check box is cleared, Smart 3D clears the inputs when you select a different batch process.

Schedule as Multiple Jobs

Processes each item in the inputs list as a separate batch process. If this check box is cleared, Smart 3D processes all of the inputs as a single batch process.

Schedule

Displays the **Schedule [Task]** dialog box so that you can schedule the batch process. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Cancel

Exits the command.

Batch Process Selection Dialog Box

Displays the controls used to specify the type of batch process to run.

Batch Process Type

Displays the general batch process types. Select a type to display the available batch process of that type in the **Select Batch Process** list.

Select Batch Process

Displays the available batch process associated with the selected type.

SECTION 34

Miscellaneous Commands

The following commands are on the **Edit** and **Tools** menus.

In This Section

Change Geometry Type (Edit Menu)..... 1385
Measure Explicit (Tools Menu) 1386
Automatic Reconnect (Tools Menu) 1388
Automatic Preview (Tools Menu)..... 1395

Change Geometry Type (Edit Menu)

Edit > **Change Geometry Type** modifies the geometry type of a plate system, a profile system, or a design seam. Use this command when you want to change a stiffener profile system by 2D projection to a stiffener profile system by table, for example.

Some geometry types are not compatible with other geometry types. You cannot change a non-linear extruded plate system to a planar plate system, for example.

You will have greater success changing geometry types of profile systems and seams, because a curve is modified. Plates systems are more difficult to change because a surface must be modified.

Modify geometry type

1. Click **Select**  on the vertical toolbar.
2. Select the appropriate filter in the **Locate Filter**: **Root Plates**, **Root Profiles**, **Seams**, or **RefCurve On Surface & Knuckles**.
3. Select the plate system, profile system, or seam to modify.
4. Click **Edit** > **Change Geometry Type**.

   The **Change Geometry Type** *dialog box displays.*
5. Slect the new geometry type, and then click **OK**.
6. Using the ribbon controls for the new geometry type, define the parameters for the object. The workflow is similar to placing a new plate system, profile system, or seam. See the appropriate topic for details:

   Place a planar plate system (on page 104)
   Place a linear extruded plate system (on page 161)
   Place a nonlinear extruded plate system (on page 185)
   Place a revolved plate system (on page 209)
   Create a ruled plate system using existing curves (on page 454)
   Place stiffener profile by intersection (on page 728)

Place stiffener profile by offset (on page 745)
Place stiffener profile by projection (on page 753)
Place stiffener profile by table (on page 760)
Place a seam by intersection (on page 903)
Place a seam by offset (on page 907)
Place a seam by projection (on page 912)
Place a seam by table (on page 914)

7. Click **Finish** on the ribbon bar to complete the geometry type change.

Measure Explicit (Tools Menu)

Measures girth distances along curves, such as landing curves, in a cumulative fashion.

The **Measure Explicit** command differs from the **Measure** command. **Measure** allows you to pick a reference point on an edge or curve and then measures the distance between that point and another point that you select on the edge/curve. The **Measure Explicit** command allows you to specify curves that intersect the edge/curve along which you are measuring. The software then computes the intersection points.

NOTES

- You can change the display of the units of measure for the distance by clicking **Tools > Options**. For more information, see *Change a Displayed Unit of Measure* in the *Common User's Guide*.
- The measurement information only appears when the command is active. This command does not write any dimensions or information to the database.

Measure Explicit Ribbon

Displays options for cumulatively measuring girth distances along curves.

Coordinate System Properties

Shows properties of the active coordinate system.

Coordinate System

Assigns the coordinate system to use to define the axis directions.

Define Coordinate System by 3 Points

Defines a temporary coordinate system by three points. Point 1 defines the origin of the coordinate system. Point 2 defines the end of the local X-axis. Point 3 defines the end of the local Y-axis.

Cumulative

Sums repeated measurements.

Clear

Resets the value of **Cumulative** to zero.

Distance

Displays the distance value. You can specify the units by clicking **Tools > Options** and selecting the **Units of Measure** tab. For more information, see *Change a Displayed Unit of Measure* in the *Common User's Guide*.

Δ X

Displays the distance along the X-axis.

Δ Y

Displays the distance along the Y-axis.

Δ Z

Displays the distance along the Z-axis.

What do you want to do?

- *Measure using a temporary coordinate system* (on page 1387)
- *Measure a girth distance along a landing curve* (on page 1387)

Measure using a temporary coordinate system

1. Click **Tools** > **Measure Explicit**.
2. On the ribbon, click **Define Coordinate System by 3 Points**  .

   The ribbon temporarily disappears.
3. Click the first point to define the origin of the coordinate system.
4. Click the second point to define the end of the local X-axis.
5. Click the third point to define the end of the local Y-axis.

   The ribbon reappears. ***Temporary*** *appears as the value in the* ***Coordinate System*** *box.*
6. Follow the status bar prompts to begin measuring.
7. Right-click to quit the **Measure Explicit** command.

Measure a girth distance along a landing curve

1. Click **Tools** > **Measure Explicit**.
2. Select the object edge to measure along.
3. Select the element from which to start the measurement. This element (curve) must intersect with the object edge that you selected in the previous step.
4. Click the second element (curve). This element (curve) must also intersect with the object edge that you selected in step 2. The girth distance from the first point to the second point appears next to the pointer.
5. Continue clicking in the model for additional measurements, as needed. The software displays the total distance in the **Cumulative** box on the ribbon.
6. Right-click to quit the **Measure Explicit** command.

Automatic Reconnect (Tools Menu)

Tools > **Automatic Reconnect** changes a boundary from one object to another. You most commonly use **Automatic Reconnect** after you copy, move, or mirror systems (including through **Model Data Reuse** and **Model Data Transform** operations in the Project Management task) without including a boundary in the copy (by selecting **Edit** > **Paste** > **Delete Optional**). These copy operations replace the boundary with a standalone plane boundary. After the copy, you can replace the standalone plane with a new boundary by using **Tools** > **Automatic Reconnect**.

You can also make other boundary changes, such as changing the boundary of a plate system from a reference plane to a plate system, or from a seam to another seam.

Automatic Reconnect Dialog Box **(on page 1391)**

What do you want to do?

- *Use filters to view copied objects required reconnection* (see "*Use filters to view copied objects requiring reconnection*" on page 1388)
- *Reconnect copied objects to a new boundary* (on page 1390)
- *Reconnect temporary definition and sketching planes to model reference planes* (on page 1390)
- *Rebound objects* (on page 1391)

Use filters to view copied objects requiring reconnection

Use this procedure to highlight objects with standalone boundaries or definition planes, resulting from a copy/move/mirror of systems, or model data reuse/model data transform in the Project Management task.

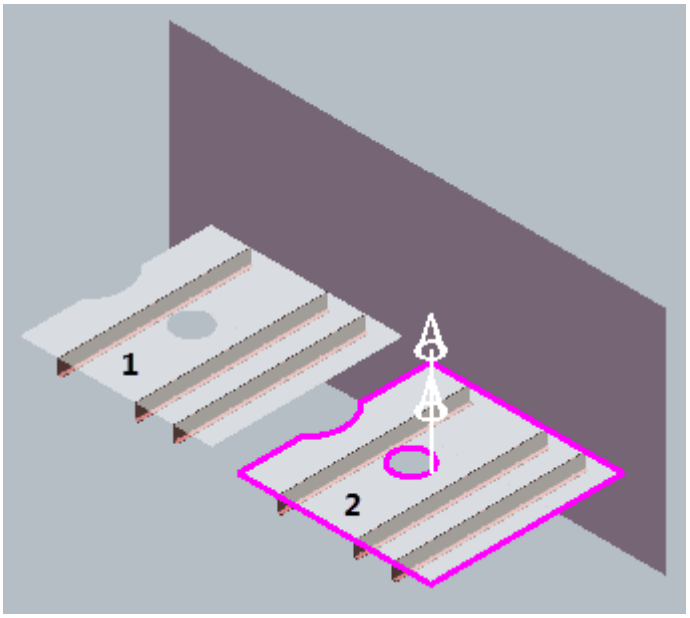
1. Click **Tools** > **Select by Filter**.

   The **Select Filter** *dialog box displays.*

2. Select one of the following:
   - Catalog Filters/Default Intelliship Filters/Intelliship Object Filters/**All Objects with Substituted Boundary**.
   - A filter in the Catalog Filters/Default Intelliship Filters/Intelliship Object Filters/All Objects with Substituted Boundary folder, such as **Plate Systems with Substituted Boundary**.
   - Catalog Filters/Default Intelliship Filters/Intelliship Object Filters/**All Objects with Substituted Definition Plane**.
   - A filter in the Catalog Filters/Default Intelliship Filters/Intelliship Object Filters/All Objects with Substituted Definition Plane folder, such as **Plate Systems with Substituted Definition Plane**.
3. Click **OK**.

   Objects meeting the filter criteria highlight in the graphic view.

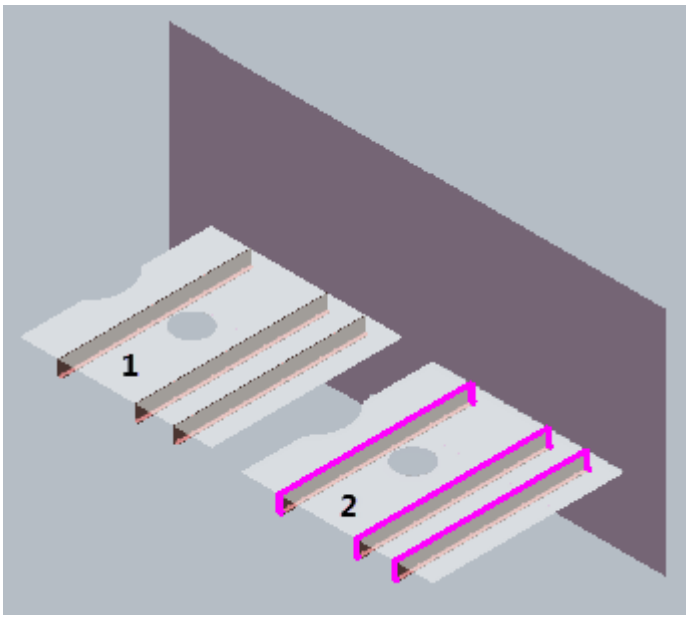
Example 1 - Plate Systems with Substituted Boundaries



1 - Original model objects

2 - Copied model objects

Example 2 - Profile Systems with Substituted Boundaries



1 - Original model objects

2 - Copied model objects

NOTE For easier access, you can copy the filters into the **My Filters** or **Ship Filters** folders in the **Select Filter** dialog box.

Reconnect copied objects to a new boundary

Use this procedure to reconnect copied objects to a boundary when the original boundary has been replaced by a temporary standalone plane as part of the copy operation.

1. Select **Tools** > **Automatic Reconnect**.

   The **Automatic Reconnect** *dialog box displays.*

2. On the **Boundary** tab, select **Reconnect**.
3. Specify the **Distance** value for finding standalone planes near the new boundary.
4. For standalone planes, select **Geometry** as the type of existing boundary.
5. For **New boundary filter**, select an object type from the list. For example, if the new boundary is a plate system, select **Plate Systems**.
6. Specify the new boundary object in the **New boundary object** box. You can select the object from either the **Workspace Explorer** or the graphic view, type the object name into the box, or select an object from the list, which shows the last ten selections you have made.
7. Optionally, to further filter restrict the objects to reconnect, select values for **Scope filter** and the **Entity filter**.
8. Click **Apply**.

   All entities that are bounded to the standalone plane display in **Candidates list** *and highlight in the graphic view.*

9. In **Candidates list**, clear the check box for objects you do not want to rebound.
10. Click **Reconnect**.
11. Click **View Log** to see the reconnect status.
12. Click **Close**.

Reconnect temporary definition and sketching planes to model reference planes

Use this procedure to reconnect copied objects to a definition or sketching plane when the original plane has been replaced by a temporary standalone plane as part of the copy operation.

1. Select **Tools** > **Automatic Reconnect**.

   The **Automatic Reconnect** *dialog box displays.*

2. On the **Plane** tab, select the type of standalone plane to reconnect, **Definition** or **Sketch**.
3. Specify the **Distance** value for finding planes near the new plane.
4. Click **Select Frame**, and select the new plane in the **Select Frame** dialog box.
5. For standalone planes, select **Geometry** as the type of existing boundary.
6. Optionally, to further filter restrict the planes to reconnect, select values for **Scope filter** and the **Entity filter**.
7. Click **Apply**.

   All objects with standalone plane definitions and within the distance value of the new plane

display in **Candidates list** *and highlight in the graphic view.*

8. In **Candidates list**, clear the check box for objects you do not want to reconnect.
9. Click **Reconnect**.
10. Click **View Log** to see the reconnect status.
11. Click **Close**.

Rebound objects

Use this procedure to replace a boundary with another boundary, such as when the boundary is used by multiple objects.

1. Select **Tools** > **Automatic Reconnect**.

   The **Automatic Reconnect** *dialog box displays.*

2. On the **Boundary** tab, select **Rebound**.
3. For **Old boundary filter**, select an object type from the list. For example, if the boundary is a reference plane, select **Grids**.
4. Specify the old boundary object in the **Old boundary object** box. You can select the object from either the **Workspace Explorer** or the graphic view, type the object name into the box, or select an object from the list, which shows the last ten selections you have made.
5. For **New boundary filter**, select an object type from the list. For example, if the boundary is a plate system, select **Plate Systems**.
6. Specify the new boundary object in the **New boundary object** box. You can select the object from either the **Workspace Explorer** or the graphic view, type the object name into the box, or select an object from the list, which shows the last ten selections you have made.
7. Optionally, to further filter restrict the objects to rebound, select values for **Scope filter** and the **Entity filter**.
8. Click **Apply**.

   All entities that are bounded to the old boundary display in **Candidates list** *and highlight in the graphic view.*

9. In **Candidates list**, clear the check box for objects you do not want to rebound.
10. Click **Rebound**.
11. Click **View Log** to see the rebound status.
12. Click **Close**.

Automatic Reconnect Dialog Box

Sets automatic rebounding parameters and performs the rebounding operation.

Boundary Tab (Automatic Reconnect Dialog Box) (on page 1392)

Plane Tab (Automatic Reconnect Dialog Box) (on page 1394)

Transactions

Single transaction

When selected, rebounds or reconnects all the selected candidates to the new boundary object in a single transaction.

Entities per transaction

Specifies the number of objects that you want to rebound or reconnect per transaction. If you do not want to commit all the objects in a single transaction, clear **Single transaction** and type the number of objects.

Plate split delay

When selected, delays the split plate update. The plate to split is placed on the **To Do List** with the message *Plate System split was stopped due to delay split* and with a status of *Out of Date*. You must select **View** > **To Do List** to update the plate system at a later time.

Rebound/Reconnect

Rebounds or reconnects the selected candidates to the new boundary object. When candidates are reconnected to the new boundary object, the software removes the candidates from the **Candidates list**.

View Log

Displays the logging information of the results on the selected candidates of the reconnect process. The **Automatic Rebound Log** dialog box displays. From this dialog box, you can click **Save As** to save the log to a file.

Close

Closes the **Automatic Reconnect** dialog box, and closes the command. If you did not click **Rebound/Reconnect**, changes are not saved.

Boundary Tab (Automatic Reconnect Dialog Box)

Sets parameters and rebounds or reconnects objects to a new boundary object.

Type

Reconnect

Restricts the command to replacing a standalone boundary resulting from a copy operation with a new object boundary. The replacement reconnects the bound plate or profile system to another system or a reference plane.

Rebound

Restricts the command to replacing a system or reference plane boundary with a new system or reference plane boundary.

Boundaries (Reconnect options)

Distance

Specifies the range distance used by the software to find existing boundaries, such as standalone planes, near the new boundary object.

New boundary filter

Specifies the object type, such as grids, seams, reference curves, plate systems, profile systems, or member systems, for the new boundary object.

NOTE Depending on the entities that you select in **Candidates list** for rebound, the

object types that do not suit as boundaries for the selected entities are available for selection.

New boundary object

Specifies the new boundary object. You can select the object from either the **Workspace Explorer** or the graphic view, type the object name into the box , or select an object from the list, which shows the last ten selections you have made.

NOTE If you select **Grids** for **New boundary filter**, you can click browse ... to display the **Select Frame** dialog box, which shows all the coordinate systems in the model. You can select the appropriate reference plane for the new boundary. For example, if you select frame F5 from coordinate system Main_CS, the **New boundary object** displays as **Main_CS:F5**.

Boundaries (Rebound options)

Old boundary filter

Specifies the object type, such as **Grids**, **Seams**, **Reference Curves**, **Plate Systems**, **Profile Systems**, or **Member Systems**, for the old boundary object.

Old boundary object

Specifies the old boundary object. You can select the object from either the **Workspace Explorer** or the graphic view, type the object name into the box, or select an object from the list, which shows the last ten selections you have made.

NOTE If you select **Grids** for **Old boundary filter**, you can click browse ... to display the **Select Frame** dialog box, which shows all the coordinate systems in the model. You can then select the appropriate reference plane for the old boundary. For example, if you select frame F5 from coordinate system Main_CS, the **Old boundary object** displays as **Main_CS:F5**.

New boundary filter

Specifies the object type, such as grids, seams, reference curves, plate systems, profile systems, or member systems, for the new boundary object.

NOTE Depending on the entities that you select in **Candidates list** for rebound, the object types that do not suit as boundaries for the selected entities are available for selection.

New boundary object

Specifies the new boundary object. You can select the object from either the **Workspace Explorer** or the graphic view, type the object name into the box , or select an object from the list, which shows the last ten selections you have made.

NOTE If you select **Grids** for **New boundary filter**, you can click browse ... to display the **Select Frame** dialog box, which shows all the coordinate systems in the model. You can select the appropriate reference plane for the new boundary. For example, if you select frame F5 from coordinate system Main_CS, the **New boundary object** displays as **Main_CS:F5**.

Candidates

Scope filter

Defines a scope to further filter the candidates list.

- **Workspace** - The software shows all the entities in the workspace that are bounded to the old boundary object.
- **System Folder** - Displays the **Select System** dialog box, where you select a system. The software shows all the entities in the selected system that are bounded to the old boundary object.
- **Select Filter** - Displays the **Select Filter** dialog box, where you select a catalog filter or create your own filter. The software shows all entities matching the filter criteria that are bounded to the old boundary object.

Entity filter

Defines an object type, such as **Plate Systems**, **Profile Systems**, **Seams**, **Reference Curves**, or **Bracket Systems**, to further filter the candidates list. You can also select **Select Filter** to create an object filter.

Candidates list

After you click **Apply**, displays all entities that are bounded to the old boundary object and satisfy the filter criteria. The candidates highlight in the graphic view. To select a candidate for rebounding, select the check box next to the candidate.

NOTE You can examine a candidate before selecting it. Click the candidate in **Candidates list**, and it displays in the system highlight color in the graphic view.

- **Description** - Displays the name of a candidate.
- **Status** - Shows the state of a candidate. The status can be **OK**, **Error**, or **To Do List**.

Apply

Populates the **Candidates list** with objects meeting the filter criteria.

Select All

Selects all the candidates available in **Candidates list**.

Select None

Clears all selections made in **Candidates list**.

Plane Tab (Automatic Reconnect Dialog Box)

Sets parameters and reconnects objects to a new reference or sketching plane.

Plane

Distance

Specifies the range distance used by the software to find systems with existing definition planes, such as the reference planes for planar plate systems, near the new reference plane.

New Planes

Select Frame

Displays the **Select Frame** dialog box, which shows all coordinate systems in the model. You can select the appropriate reference plane for the new plane. For example, if you select frame Z1 from coordinate system Main_CS, the **New boundary object** displays as **Main_CS:Z1**.

New Planes

Displays the new reference plane.

Candidates

Scope filter

Defines a scope to further filter the candidates list.

- **Workspace** - The software shows all the entities in the workspace that are defined by a standalone plane and within the range distance of the new plane.
- **System Folder** - Displays the **Select System** dialog box, where you select a system. The software shows all the entities in the selected system that are defined by a standalone plane and within the range distance of the new plane.
- **Select Filter** - Displays the **Select Filter** dialog box, where you select a catalog filter or create your own filter. The software shows all entities matching the filter criteria that are defined by a standalone plane and within the range distance of the new plane.

Entity filter

Defines an object type, such as **Plate Systems**, **Profile Systems**, **Seams**, **Reference Curves**, or **Bracket Systems**, to further filter the candidates list. You can also select **Select Filter** to create an object filter.

Candidates list

After clicking **Apply**, displays all entities that are defined by a standalone plane, within the range distance of the new plane, and satisfy the filter criteria.The candidates highlight in the graphic view. To select a candidate for reconnection, select the check box next to the candidate.

NOTE You can examine a candidate before selecting it. Click the candidate in **Candidates list**, and it displays in the system highlight color in the graphic view.

- **Description** - Displays the name of a candidate.
- **Status** - Shows the state of a candidate. The status can be **OK**, **Error**, or **To Do List**.

Apply

Populates the **Candidates list** with objects meeting the filter criteria.

Select All

Selects all the candidates available in **Candidates list**.

Select None

Clears all selections made in **Candidates list**.

Automatic Preview (Tools Menu)

Tools > **Automatic Preview** sets advanced plate system and geometric construction palettes to automatically display previews for a selected macro. For more information, see *Advanced Plate Palette Dialog Box* (on page 283) and *Geometric Constructions* (on page 1027).

APPENDIX A

Swap Hulls on a Detailed Model Using Molded Forms Delay

Use this workflow when a ship is far enough into the design cycle that changing the hull form causes recalculation of a large number of objects. This workflow uses settings on the **Delay** tab of the **Imported Plate System Properties** dialog box, allowing changed objects to be better managed to prevent model errors and excessive recalculation time. The following types of recalculations are delayed:

- Hull splits
- Update of hull structure, such as hull profiles
- Update of internal structure dependent on the hull

★IMPORTANT

- This workflow is best performed by an administrator with full permissions to the model.
- This workflow is best performed on a 64-bit Windows computer with at least 4 GB of RAM. For more information, see *Intergraph Smart™ 3D Installation Guide*.
- If your model is very early in the design process, you may be able to use the basic hull swap procedure described in *Swap hulls* (on page 137). If you are unsure of which procedure to use, follow the workflow below.

What do you want to do?

- *Prepare the model* (on page 1397)
- *Copy the model* (on page 1397)
- *Start error logging* (on page 1398)
- *Select delay and range settings and swap the hulls* (on page 1398)
- *Check the To-Do List* (on page 1400)
- *Check the seam pattern on the new hull* (on page 1400)
- *Repair seams that did not split* (on page 1401)
- *Remove delay and update To-Do List records for the hull* (on page 1402)
- *Check hull records on the To-Do List* (on page 1402)
- *Remove delay and update To-Do List records for internal structure* (on page 1403)
- *Check internal structure records on the To-Do List* (on page 1403)
- *Check internal structure in the model* (on page 1403)
- *Update remaining To-Do List records* (on page 1403)
- *Evaluate results* (on page 1404)

- *Recover from a failed hull swap* (on page 1405)
- *Stop error logging* (on page 1405)

Prepare the model

The following steps are recommended to ensure that the model is in the cleanest possible state before swapping the hull.

1. In Project Management, click **Database** > **Integrity** to scan the model for records that may cause performance problems and can be removed, such as records of deleted objects. For more information, see *Database Integrity* in the *Project Management User's Guide*.
2. In Project Management, click **Database** > **Maintenance** to clean the unneeded records from database. For more information, see *Database Maintenance* in the *Project Management User's Guide*.
3. In SQL, run the following queries:
   - *[Product Folder]*\StructDetail\Server\Schema\SQL\DataIntegrityScripts\FixDanglingFlavorRlns.sql
   - *[Product Folder]*\StructDetail\Server\Schema\SQL\DataIntegrityScripts\CleanUnnecessaryFlavors.sql
4. In the model, click **Tools** > **Data Integrity Check.** Perform the check, and fix listed items related to seam gaps and seams on the To Do List. For more information, see *Check data integrity of plates* in *Data Integrity Check (Tools Menu)* (on page 1371).
5. Optionally, export the seam data by clicking **File** > **Export** > **Seams**. This seam data can be imported onto the new swapped hull if problems occur during the swap. For more information, see *Export Seams (File Menu)* (on page 1239).
6. If delay is used on any existing model objects, Click **Tools** > **Delay Settings** and update all structure. The steps are similar to *Remove delay and update To-Do List records for internal structure* (on page 1403).
7. Click **View** > **To Do List**. Fix as many listed records as possible.

Copy the model

1. Create a backup of the databases. For more information, see *Backup* in the *Project Management User's Guide*.
2. From the backup, restore a copy of the model into the existing site. Give the copied model a different name. For more information, see *Restore* in the *Project Management User's Guide*.

   NOTE The copied model is later used as the before-swap model to compare with the after-swap model. For more information, see *Evaluate results* (on page 1404).
3. Open the original model.

Start error logging

1. Open **EnableErrorLog**. EnableErrorLog.exe is delivered in *[Product Folder]*\Core\Tools\Administrator\Bin.

   *The **Enable Error Log** window opens.*

2. Select **Log run time errors**.
3. Select **2 - Detailed** for **Select the severity of the errors to be reported**.
4. If you do not want to use the default location and file name, type a new path for **Error log full file name**.
5. Click **Generate time-based log file name at each new start of SP3D task host** and **Preserve User Settings**.
6. Click **OK**.

   *The **Enable Error Log** window closes. Error logging runs in the background when Smart 3D is running.*

Select delay and range settings and swap the hulls

1. Click **Select** on the vertical toolbar.
2. Select **Molded Forms (Root Level)** in the **Locate Filter**.
3. Select the existing hull plate system in a graphic view or the Workspace Explorer.

   *The **Imported Plate System** ribbon appears.*

4. Click **File Definition**.

   *The **Open Hull File** dialog box appears.*

5. Select the .SAT or .IGS file that contains the new hull to import.

   IMPORTANT

   - The names, types, and attributes of reference curves and knuckles on the new hull must match those of the old hull. Unmatched reference curves and knuckles are deleted.
   - To show .IGS files in the file browser, you must select **IGES files (*.igs)** as the file type.

6. Click **Open**.

   A preview of the new hull appears in the graphic view.

7. If you are importing a hull from an .IGS file and the file defines only half of the hull, click **Mirror surface**, **Mirror reference curves**, or both on the ribbon to import a complete hull.

   NOTE By default, the two mirror options are unavailable. A preview for the hull is updated automatically when you click **Mirror surface**.

8. On the **Imported Plate System** ribbon, click **Imported Plate System Properties**.

   *The **Imported Plate System Properties** dialog box appears.*

For more information, see *Imported Plate System* (on page 130) and *Delay Tab (Imported Plate System Properties Dialog Box)* (on page 155).

9. On the **Delay** tab, select **Molded Forms Delay**.
10. Optionally, select **Plate Split Delay**.

    TIP This option provides the fastest performance for the hull swap.
11. If you do not know the coordinate range differences between the old and new hull and want all hull-related recalculations to be delayed, then select **System range of change**.

    Low *and* ***High*** *values are added in the* ***Display Range*** *grid.*
12. If you know the coordinate range differences between the old and new hull or only want to delay recalculations in one or more limited areas:
    - Select **User range of change**.
    - Type **Low** and **High** values for the **X**, **Y**, and **Z** axes.
    - Click **Add**.

    The values appear in the ***Display Range*** *grid.*
13. Repeat the previous steps to create additional ranges as needed.
14. To check a range, select a row in the **Display Range** grid.

    A range box appears in the graphic view.

    NOTE When **System range of change** is selected, verify that the range box includes the entire old and new hulls.
15. If the range box is not correct, type the needed **Low** and **High** values for **X**, **Y**, and **Z**, then click **Modify**.
16. Click **OK**.

    The ***Imported Plate System Properties*** *dialog box closes.*
17. Click **Finish** on the ribbon.

    The old hull is replaced with the new hull. Hull seams, structure, and reference curves are moved to the new hull.

Check the To-Do List

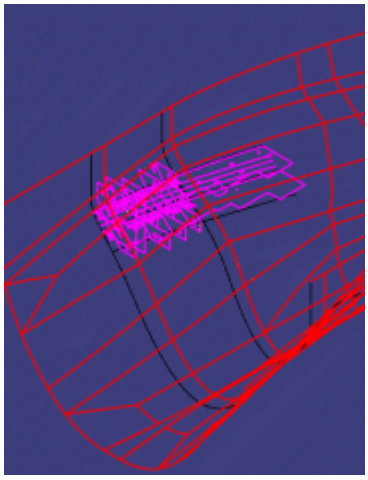
1. Click **View** > **To Do List**.

   *The **To Do List** dialog box appears.*

2. Select records with the following messages:

   - Compute was stopped due to delay propagation.
   - Plate System bound was stopped due to delay propagation.
   - Landing curve bound was stopped due to delay propagation.
   - Plate system split was stopped due to delay split during molded forms delay.
   - Bracket re-symbolization was stopped due to delay propagation.
   - Plate part generation was stopped due to delay propagation.
   - Profile part generation was stopped due to delay propagation.
   - One or more connected leaf systems are delayed. Please update the connected object in delay.

   The affected objects highlight in the graphic view. In the example below, internal structure with the "Plate System bound was stopped due to delay propagation" message is selected.

3. Check that the objects are in the range boxes that you defined on the **Delay** tab.

Check the seam pattern on the new hull

1. Add a **Verify Seam Pattern to Split** custom command for ProgID **SplitCommands.VerifySeamPatternToSplit**. For more information, see *Custom Commands* in the *Common User's Guide*.
2. Click **Select** on the vertical toolbar.
3. Select **Molded Forms (Root Level)** in the **Locate Filter**.
4. Select the hull plate system.

5. Click **Tools** > **Custom Commands**.

   *The **Custom Commands** dialog box appears.*

6. Select **Verify Seam Pattern to Split** in the **Command name** box and click **Run**.

   A dialog box appears listing:

   - *Successful and failed splits (bad splitters).*
   - *The number of modified, new, identical, and deleted leaf systems.*

7. Select options in the dialog box.

   *The selected options display in the graphic view. In the following example, **Modified Bodies (Yellow)** is selected. Modified leaf systems display in yellow.*



8. Check the results.

Repair seams that did not split

Perform these steps if any seams failed to split on the new hull.

1. Click **Tools** > **Data Integrity Check.**
2. Perform the check. For more information, see *Check data integrity of plates* in *Data Integrity Check (Tools Menu)* (on page 1371).
3. In the **Results** dialog box, select **Enable filters**. Select **Knuckles** and **Seams** in the **Filter selection** box. For more information, see *Results Dialog Box* (on page 1372).
4. Try to fix the listed items according to the help provided with **Data Integrity Check**.
5. Repeat these steps until all bad splits are fixed.

NOTE If you are certain that items with a **Warning** status do not have an impact on the model, then you can ignore them.

Remove delay and update To-Do List records for the hull

1. Click **Tools** > **Delay Settings**.

   *The **Delay Operations** dialog box appears.*

2. On the **Delay Settings** tab, click **No Delay**, and then click **Apply**.
3. On the **Update To Do Records** tab, select **Optimal**.
4. Select **Out-of-Date Molded Forms**. Clear all other **Type** options.
5. For **Structure to Update**, select or type the name of the hull.

   *The number of delayed objects to update is shown in the **Update To Do Records** frame.*

6. Select **Enable Logging**.
7. Click **Update Objects**.

   Updates of hull records on the To Do List are performed by the software. Internal structure is still delayed. For more information, see Optimal Update Processing for Hull Swaps *(on page 1406).*

8. Click **View Log File** and check results of the update.
9. Click **OK**.

For more information, see *Delay Settings (Tools Menu)* (on page 1366) and *Delay Operations Dialog Box* (on page 1367).

Check hull records on the To-Do List

1. Click **View** > **To Do List**.

   *The **To Do List** dialog box appears.*

2. Select records with "delay propagation" messages.

   The delayed objects highlight in the graphic view.

3. Check that:
   - The objects are in the range boxes that you defined on the **Delay** tab.
   - Most remaining records are for internal structure.
   - Remaining hull records are valid. For example, a hull profile bounded or split by internal structure is still out-of-date if the internal structure is out-of-date.
4. Fix or update remaining hull records, as described in *Repair seams that did not split* (on page 1401) and *Remove delay and update To-Do List records for the hull* (on page 1402).

Remove delay and update To-Do List records for internal structure

1. Click **Tools** > **Delay Settings**.

   *The **Delay Operations** dialog box appears.*
2. On the **Delay Settings** tab, click **No Delay**, and then click **Apply**.
3. On the **Update To Do Records** tab, select **Optimal**.
4. Select **Out-of-Date Molded Forms**. Clear all other **Type** options.
5. For **Structure to Update**, select **All**.

   *The number of delayed objects to update is shown in the **Update To Do Records** frame.*
6. Select **Enable Logging**.
7. Click **Update Objects**.

   Updates of remaining delay records on the To Do List are performed by the software. For more information, see Optimal Update Processing for Hull Swaps *(on page 1406).*
8. Click **View Log File** and check results of the update.
9. Click **OK**.

For more information, see *Delay Settings (Tools Menu)* (on page 1366) and *Delay Operations Dialog Box* (on page 1367).

Check internal structure records on the To-Do List

1. Click **View** > **To Do List**.

   *The **To Do List** dialog box appears.*
2. Select records with "delay propagation" messages.

   The delayed objects highlight in the graphic view.
3. Fix or update remaining delay records.

Check internal structure in the model

Perform the following checks in the graphic view and the Workspace Explorer.

1. Check connections for internal structure that were affected by the hull swap.
2. Check detailing objects for plates and profiles that were affected by the hull swap.

Update remaining To-Do List records

1. Click **Tools** > **Delay Settings**.

   *The **Delay Operations** dialog box appears.*
2. On the **Delay Settings** tab, click **No Delay**, and then click **Apply**.

3. On the **Update To Do Records** tab, select **Optimal**.
4. Select **Errors**, **Warnings**, and **Out-of-Date** for **Type**.
5. For **Structure to Update**, select **All**.

   *The number of objects to update is shown in the **Update To Do Records** frame.*
6. Select **Enable Logging**.
7. Click **Update Objects**.

   Updates of remaining records on the To Do List are performed by the software.
8. Click **View Log File** and check results of the update.
9. Click **OK**.
10. Click **View** > **To Do List**.

    *The **To Do List** dialog box appears. Remaining items on the To Do List are likely modeling errors that should be fixed.*

    For more information, see Delay Settings (Tools Menu) *(on page 1366) and* Delay Operations Dialog Box *(on page 1367).*

Evaluate results

1. Add an **Audit Tool** custom command for ProgID **stAnalysisTool.AuditTool**. For more information, see *Custom Commands* in the *Common User's Guide*.
2. Click **Tools** > **Custom Commands**.

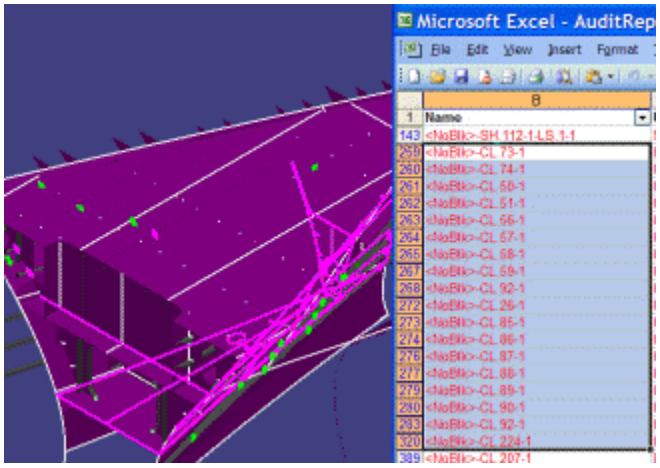
   *The **Custom Commands** dialog box appears.*
3. Select **Audit Tool** in the **Command name** box and click **Run**.

   *The **Audit Report** dialog box appears.*
4. Click **Base Model**.

   *The **Set Connection String** dialog box appears.*
5. For **Database**, select the server and copied model database name created in *Copy the model* (on page 1397). This is a copy of the current model before the hull swap.
6. For **Metadata**, select the server and catalog schema for the copied model database. This is the same catalog schema as the current model.
7. Click **OK**.
8. In the **Audit Report** dialog box, select **Show objects in current workspace** (under **Report Option**) and **Highlight objects in both models** (under **Visualize**).
9. Define a comparison start data and time, and click **Process**.

   An audit report created in a .xls workbook opens after processing completes.
10. Select one or more rows in the .xls workbook.

Modified objects are selected in the graphic view.



11. NOTE Audit reports are saved to the C:\\temp folder. Click **Load** on the **Audit Report** dialog box to open an existing audit report and view the results at any time.

Recover from a failed hull swap

Perform the following steps if the hull swap fails.

1. Copy the log files created in *Start error logging* (on page 1398), *Remove delay and update To-Do List records for the hull* (on page 1402), and *Remove delay and update To-Do List records for internal structure* (on page 1403) to a different location so that they are available for future analysis.
2. Repeat the steps in *Remove delay and update To-Do List records for the hull* (on page 1402), *Remove delay and update To-Do List records for internal structure* (on page 1403), and *Update remaining To-Do List records* (on page 1403) until all delay-related To Do List records are removed.

NOTE When the software recalculates an object that failed on the previous attempt, you are asked if you want to process it again.

Stop error logging

1. Open **EnableErrorLog**.

   *The **Enable Error Log** window opens.*

2. Clear **Log run time errors**.
3. Click **OK**.

   *The **Enable Error Log** window closes. Error logging no longer runs in the background when Smart 3D is running.*

Optimal Update Processing for Hull Swaps

With **Tools** > **Delay Settings**, you can define options on the **Delay Settings** tab that optimize update processing. In the following example workflow, hull structure is updated first, followed by internal structure.

First Update

The following update sequence is used when **Optimal** and **Out-of-Date Molded Forms** are selected, and the hull is defined for **Structure to Update**. The software repeatedly updates until all delayed hull objects are processed.

1. Hull root plate system is split, with all new or modified leaf systems delayed.
2. Hull root profile systems are updated, with internal structure dependent on the profiles delayed.
3. Hull leaf plate systems are updated, with internal structure dependent on the hull delayed.
4. Hull logical connections are updated, with internal structure dependent on the connections delayed.
5. Hull plate parts are updated, with internal structure dependent on the parts delayed.
6. Hull detailing objects are updated, with internal structure dependent on the objects delayed.

Second Update

The following update sequence is used when **Optimal** and **Out-of-Date Molded Forms** are selected, and **All** is defined for **Structure to Update**. The software repeatedly updates until all remaining delayed objects are processed.

1. Internal root plate systems are updated, with all leaf systems and dependent structure delayed.
2. Leaf plate systems are updated, with dependent structure delayed.
3. Profile systems are updated, with dependent structure delayed.
4. Bracket systems are updated, with dependent structure delayed.
5. Logical connections are updated, with dependent structure delayed.
6. Plate and profile parts are updated, with dependent structure delayed.
7. Detailing objects are updated, with dependent structure delayed.

For more information, see *Delay Settings (Tools Menu)* (on page 1366).

APPENDIX B

Data Integrity Check Messages

This appendix describes the messages and possible solutions for the **Results** dialog box of the **Data Integrity Check** command in the Molded Forms task. For more information, see *Data Integrity Check (Tools Menu)* (on page 1371).

In This Appendix

All seams or knuckles are ignored in split. 1408
Bad connection (either missing port or invalid port in connection) 1408
Bad edge attributes because of knuckle <name of knuckle>. Check connection. 1408
Bad edge attributes because of seam <name of seam>. Check connection. 1409
Connection has no geometry. 1409
Could not find the boundary/operator creating the edge. Check the connection. 1410
Could not get back the edge information. Check connection 1410
Could not retrieve boundary. Check connection. 1410
Could not retrieve edge information. Check connection. 1411
Could not retrieve the children of the plate system. 1411
Could not retrieve the root plate. 1411
Duplicate Physical Connection. 1412
Edge port doesn't have geometry. Check connection. 1412
Entire length of seam ignored in split. 1413
Error - Has opposite surface normal as the root plate <name of root plate> 1413
Failure encountered while checking seam geometry and split result. 1413
Failure encountered while retrieving thickness. 1414
Gap detected in the seam bounding. 1414
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All seams or knuckles are ignored in split.

Description

No seams or knuckles are in the split list of the plate.

Possible Cause

The **Execute Split** command has not been carried out. This message is expected if **Data Integrity Check** is run before a plate is split.

What You Should Do

Select the plate, and carry out the **Execute Split** command.

Bad connection (either missing port or invalid port in connection)

Description

A root logical connection has a missing or invalid destination port.

Possible Cause

Unknown.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Bad edge attributes because of knuckle <name of knuckle>. Check connection.

Description

The geometry of the knuckle does not overlap the corresponding plate edge. As a result, the edge attributes are incorrect.

Possible Cause

Bad bounding or split operation.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. If the knuckle uses sketched geometry, extend it past the plate edges.
4. Click **Recompute** in the **Results** dialog box to recalculate the object.

Bad edge attributes because of seam <name of seam>. Check connection.

Description

The geometry of the seam does not overlap the corresponding plate edge. As a result, the edge attributes are incorrect.

Possible Cause

Bad bounding or split operation.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. If the seam uses sketched geometry, extend it past the plate edges.
4. Click **Recompute** in the **Results** dialog box to recalculate the object.

Connection has no geometry.

Description

Geometry of the logical connection cannot be read.

Possible Cause

Missing or corrupt root logical connection.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Could not find the boundary/operator creating the edge. Check the connection.

Description

An edge input was not found during inspection of the plate edges.

Possible Cause

The object responsible for creating the edge on the plate no longer exists or is corrupted.

What You Should Do

Click **Recompute** in the **Results** dialog box to recalculate the object.

Could not get back the edge information. Check connection

Description

An error occurred while reading the edge attributes of the object.

Possible Causes

Corrupted object.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Could not retrieve boundary. Check connection.

Description

An error occurred while reading the boundary attributes of the object.

Possible Causes

A boundary no longer exists or a boundary is corrupt.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.

2. Manually correct other objects with errors associated with the object.
3. Rebound the object.

Could not retrieve edge information. Check connection.

Description

Unable to find the plate edges.

Possible Cause

Corrupt plate.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Could not retrieve the children of the plate system.

Description

No child plate system could be retrieve from the root plate system during inspection. Every root plate system should have at least one child.

Possible Cause

Error during a split.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Could not retrieve the root plate.

Description

Object is the root system of another object, but could not be read as a root system.

Possible Causes

Corrupted object.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Duplicate Physical Connection.

Description

Two overlapping physical connections exist in the model.

Possible Cause

A manual physical connection is placed in a location where physical connections are generated by rule.

What You Should Do

Delete the manually-placed physical connections.

Edge port doesn't have geometry. Check connection.

Description

Error while getting the geometry of an edge port.

Possible Cause

Computation error during plate modification.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.
4. Delete the object and recreate it.

Entire length of seam ignored in split.

Description

The seam was included in the split list of the plate, but no part of it was used as part of the split.

Possible Cause

Corrupt or out-of-date seam.

What You Should Do

Separately try each of the following steps:

1. Click **Recompute** in the **Results** dialog box to recalculate the seam.
2. Delete the seam and recreate it.

Error - Has opposite surface normal as the root plate <name of root plate>

Description

The root plate and child plate surface normals differ.

Possible Cause

Computation error during propagation in a split operation.

What You Should Do

Separately try each of the following steps:

1. Select the child plate system. Use **Tools** > **Recompute Objects** to recalculate the object.
2. Select the root plate system. Use **Tools** > **Recompute Objects** to recalculate the object.

Failure encountered while checking seam geometry and split result.

Description

An unknown error occurred while the analysis tool was checking the seams and geometry on a plate

Possible Cause

Corrupted seam geometry

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.
4. Delete the object and recreate it.

Failure encountered while retrieving thickness.

Description

Plate thickness was not retrieved during inspection.

Possible Cause

Corrupt plate system.

What You Should Do

Separately try each of the following steps:

1. Check the plate system properties to verify that the properties, especially the material and thickness, are set properly.
2. Click **Recompute** in the **Results** dialog box to recalculate the object.

Gap detected in the seam bounding.

Description

A gap exists between a seam landing curve and one of its boundaries.

Possible Cause

1. A sketched seam does not intersect its boundary.
2. The seam is out-of-date.

What You Should Do

Separately try each of the following steps:

1. Select the row in the **Results** dialog box. The beginning of the gap is shown in the graphic view with this icon:  .
2. Open the seam in **Sketch 2D** and verify that the seam landing curve extended past its boundaries.
3. Click **Recompute** in the **Results** dialog box to recalculate the seam.

In to-do list: <to-do list item>

Description

The object has one or more to-do list items associated with it

Possible Cause

Computation error.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Inconsistent attributes between system and part geometry. Check connection.

Description

Plate part wire geometry does not fit entirely within the plate system wire geometry, as expected.

Possible Causes

1. Improper modification of the plate part and/or system.
2. The part or system was corrupted during database migration/upgrade.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.
4. Delete the object and recreate it.

Inconsistent port attributes between root plate and leaf plate. Check connection.

Description

The port attributes between the logical connection of the root and leaf plates do not match.

Possible Cause

Missing or corrupt root logical connection.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Knuckle <name of knuckle> has no geometry.

Description

No geometry was found for the knuckle.

Possible Cause

Corrupt or out-of-date knuckle.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

<length in meters> of seam length ignored in split.

Description

The seam was included in the split list of the plate, but part of the length was not used as part of the split.

Possible Cause

Computation error.

What You Should Do

Separately try each of the following steps:

1. Select the row in the **Results** dialog box. The point where the length is ignored is shown in the graphic view with this icon:  .
2. Click **Recompute** in the **Results** dialog box to recalculate the object.

Missing boundary connection to <object name>

Description

The object specified in the error message bounds the plate but no bounding logical connection for it exists.

Possible Cause

Missing or corrupt root logical connection.

What You Should Do

Separately try each of the following steps:

1. Click **View** > **To Do List**. Select any items associated with the root logical connection. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Missing child connections.

Description

The number of child logical connections on the plate is less than the number of splitters intersecting the plate.

Possible Cause

Missing or corrupt root logical connection.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Missing connection with <object name>

Description

An edge on this plate and another plate overlap and should therefore have a logical connection between them, but the connection is missing.

Possible Cause

Missing or corrupt root logical connection.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Missing root intersection connection with <object name>

Description

The object specified in the error message splits and intersects the plate but no intersection connection for it exists.

Possible Cause

Missing or corrupt root logical connection.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

No splitting surface - will be ignored in split. Check the seam creation.

Description

The splitting surface attribute was not found on the seam. Because of this, it is not used in splitting the plate.

Possible Cause

Corrupt seam

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Not used in splitting.

Description

The seam is on a plate that is split, but it was not added to the split list

Possible Cause

The seam was created after the plate was split or it was not selected when the **Execute Split** command was run on the plate.

What You Should Do

1. From the **Execute Split** command, add the seam to the split list and apply the changes.

Seam <name of seam> has no geometry.

Description

No geometry detected for the seam.

Possible Cause

Corrupt or out-of-date seam.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Seam overlaps knuckle <name of knuckle>

Description

The landing curve of the knuckle overlaps that of another knuckle. There may be problems when splitting.

Possible Cause

The boundaries of the seam are not set properly or the seam has the wrong solution chosen if it is ambiguous.

What You Should Do

Separately try each of the following steps:

1. Select the row in the **Results** dialog box. The point where the two affected seams overlap is shown in the graphic view with this icon:  .
2. Verify that the two seams have the right boundaries chosen and if the seam is ambiguous, verify that the right solution is chosen.
3. Click **Recompute** in the **Results** dialog box to recalculate the seam.

Seam overlaps seam <name of seam>.

Description

The landing curve of the seam overlaps the landing curve of another seam. This creates potential issues when attempting to split.

Possible Cause

The boundaries of the seam are not correct or the seam has the wrong solution chosen if it is ambiguous.

What You Should Do

Separately try each of the following steps:

1. Select the row in the **Results** dialog box. The point where the two affected seams overlap is shown in the graphic view.
2. Verify that the two seams have the right boundaries chosen and if the seam is ambiguous, verify that the right solution is chosen.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Part light part geometry might be corrupted.

Description

Geometry of the object could not be read.

Possible Cause

Corrupted data.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Plate has no plate part children.

Description

The child plate system does not have a plate part.

Possible Cause

Corrupt plate part or child plate system. Computation error during propagation in a split operation.

What You Should Do

Separately try each of the following steps:

1. Select the child plate system. Click **Recompute** in the **Results** dialog box to recalculate the object.
2. Select the root plate system. Click **Recompute** in the **Results** dialog box to recalculate the object.

The seam has multiple segments; there may be a problem in the split.

Description

Seams with multiple segments may not valid.

Possible Cause

An ambiguous solution may not have been solved correctly, leaving two or more seam landing curves.

What You Should Do

Seams composed of multiple segments are acceptable when they form an unbroken line across the plate to be split. Otherwise, this indicates a problem with the seam bounding. Verify that the proper seam boundaries are selected and the proper solution is selected if the seam is ambiguous.

It may be possible to correct this issue with the **Recompute** command. Click **Recompute** in the **Results** dialog box to recalculate the object.

The wrong item is used in the bounding logical connection.

Description

The geometry of the destination port used in the bounding logical connection is not actually bounding the plate.

Possible Cause

Corrupt or out-of-date root logical connection.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

There is a gap in the boundary <name of boundary>. Detailing might fail.

Description

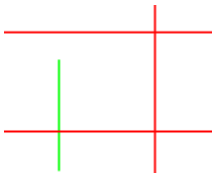
An error was encountered while inspecting the specified object. A gap exists between the edge of the object and the boundary. This can cause problems in the Structural Detailing task.

Possible Cause

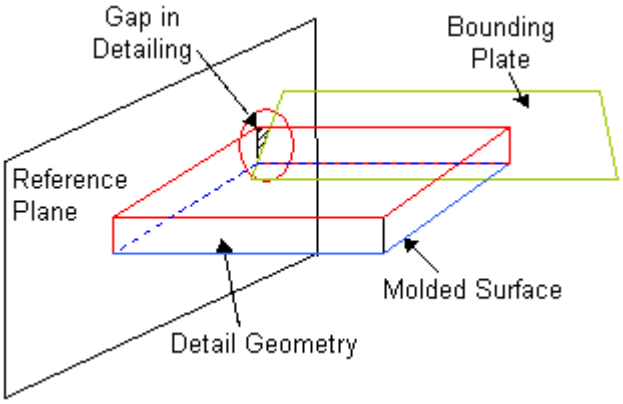
The object is using infinite geometry as a boundary.

The free edge gap can be caused by one of two situations:

- Unbounded geometry is used for the plate bounding. For example, create a plate bounded by 3 reference planes (red) and 1 planar plate (green). Because of the infinite geometry of the bounding plate is used, the plate is created successfully in Molded Forms. But the plate will fail in detailing because there is a gap in the trim boundary caused by the bounding plate.



- Incomplete trim boundary due to the plate thickened geometry. In Molded Forms, the boundary is complete, but in detailing, there is a small gap along the thicken direction. In most cases, the gap is very small.



What You Should Do

Check and update the object boundaries.

Too many child connections.

Description

There are more child logical connections than splitters intersecting the plate.

Possible Cause

Missing or corrupt root logical connection.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.

Unable to check for seams in To Do List.

Description

An error was encountered while checking for seams on the to-do list.

Possible Cause

Computation error with the to-do list.

What You Should Do

Contact Intergraph Support for assistance.

Unknown error

Description

An unexpected error was encountered while inspecting the specified object.

Possible Cause

Unknown.

What You Should Do

Separately try each of the following steps:

1. Right-click and select **Update**.
2. Manually correct other objects with errors associated with the object.
3. Click **Recompute** in the **Results** dialog box to recalculate the object.
4. Delete the object and recreate it.

Warning!!--Possible wrong thickness direction among leaf parts.

Description

A child plate system was found that did not match one of the other child plate systems under the same parent plate system.

Possible Cause

Computation error during propagation in a split operation.

What You Should Do

Separately try each of the following steps:

1. Select the child plate system. Use **Tools** > **Recompute Objects** to recalculate the object.
2. Select the root plate system. Use **Tools** > **Recompute Objects** to recalculate the object.

APPENDIX C

Tripping Bracket

Tools > **Tripping Bracket** is provided *only* for compatibility with brackets created with the old **Bracket Plate System** command in earlier versions of the software.

To create new, similar brackets, use **Bracket Plate System** and select **Plane by Selected Supports** as the value for **Select Plane**. For more information, see *Bracket Plate System* (on page 227).

Tripping Bracket Ribbon

Displays the options that you use to place a bracket plate system in the model.

Properties

Opens the **Bracket Plate System Properties** dialog box, which you use to view and modify the properties of the bracket system that you are about to place in the model. The initial bracket system properties default from the selected structural specification. For more information, see *Bracket Plate System Properties Dialog Box* (on page 1429).

Select bracket and flange symbols

Specifies the type of bracket system to place. Also optionally specifies the type of flange to place on the bracket system.

Bracket Symbol

Select **More** to select a bracket symbol from the catalog. Select **Sketched** to sketch the free edge of the bracket.

Flange Symbol

Select **More** to select a flange symbol from the catalog. Select **No Flange** if the bracket does not need a flange.

★IMPORTANT Only use a flange symbol with a bracket symbol containing parameters for the addition of a flange.

Flip Flange

Flips the orientation of the flange about an axis normal to the bracket free edge. If the flange is symmetrical, **Flip Flange** has no visible effect.

Type

Select the type of bracket system to place. The bracket type is classified by the number of supports.

Select Support 1

Select the first support for the bracket plate system.

Length 1

Enter a value to define a point on the bracket plane.

- If the support is linear, then the **Length 1** value can be any number greater than zero.
- If the support is curved, then the **Length 1** value can be adjusted to change the bracket position relative to the support.

Select Support 2

Select the second support for the bracket plate system.

Length 2

Enter a value to define a second point on the bracket plane.

- If the support is linear, then the **Length 2** value can be any number greater than zero.
- If the support is curved, then the **Length 2** value can be adjusted to change the bracket position relative to the support.

Select Support 3

Select the third support for the bracket plate system. (Available only for 3S, 4S, and 5S brackets.)

Select Support 4

Select the fourth support for the bracket plate system. (Available only for 4S and 5S brackets.)

Select Support 5

Select the fifth support for the bracket plate system. (Available only for 5S brackets.)

Constraint

Select the method by which the third point of the bracket plane is defined. This option is unavailable until **Select Support 1** and **Select Support 2** have been defined.

- **Parallel** - Bracket plane is parallel to the support 1 plane.
- **Perpendicular** - Bracket plane is perpendicular to the support 1 plane.
- **Transverse** - Bracket plane is transverse.
- **Vertical** - Bracket plane is vertical.
- **Fixed X** - Rotate Bracket around the fixed X-Axis.
- **Fixed Y** - Rotate Bracket around the fixed Y-Axis.
- **Fixed Z** - Rotate Bracket around the fixed Z-Axis.

Fixed Value

Value of fixed axis rotation. This option appears after one of the fixed axis third point locations have been selected.

Resolve ambiguity/sketch boundary

Switches to the Sketch 2D commands when creating a sketched bracket.

Bracket Reinforcement

Adds a reinforcement to the bracket.

Sketching Plane

Specify the sketching plane for the curve. This is the first step in defining the curve.

Add Intersecting Item

Allows you to select objects that intersect the sketching plane in the 3D environment. You see the selected objects in the Draft 2D environment when you are drawing the opening.

Add Projection Item

Allows you to select objects that do not intersect the sketching plane in the 3D environment. The objects are projected onto the sketching plane, and you see the selected objects in the Draft 2D environment when you are drawing the opening.

Finish

Places the bracket plate system in the model.

Auto Finish Mode

Click to automatically place the bracket plate system in the model as soon as you set all of the required inputs. When set to **Off**, you must click **Finish** to place the system.

Sketch 2D

Opens the Sketch 2D environment in which you can draw the curve.

NOTE To display a grid or reference plane ruler in the Sketch 2D environment, first define an active coordinate system using **Tools** > **PinPoint**. For more information, see PinPoint in the *Common User's Guide* and *2D Grids Ruler* in the *SmartSketch Drawing Editor Help*.

Auto Import Enable/Disable

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Project Item** option. For more information, see *Sketch 2D Best Practices* (on page 65).

Override Offsets

Ignores the rule-based U and V offsets for all three points that are used to define the plane of the tripping bracket, and sets the offsets to zero and the override flag to **True**.

Accept

Accepts the active selection or state.

L/T

Displays the length divided by the thickness. This value is used by the rules to calculate the reinforcement and cannot be edited.

By Rule

Turn on the checkbox to let the rules calculate an appropriate reinforcement. Turn off the checkbox to manually define the reinforcement.

Reinforcement

Displays the reinforcement type. You can change the reinforcement type if the **By Rule**

checkbox is turned off.

Properties

Activates the **Bracket Reinforcement Properties** dialog box, which you use to view and modify the properties of the bracket reinforcement that you are about to place in the model. The initial reinforcement properties default from the selected bracket and the bracket reinforcement rules. For more information see *Bracket Reinforcement Properties Dialog Box* (on page 271).

Section Size

Select the section size for the reinforcement. This is read-only if the **Override Cross Section Information** check box is turn off on the **Bracket Reinforcement Properties** dialog box. For more information, see *Cross Section Tab (Bracket Reinforcement Properties Dialog Box)* (on page 272).

Primary Orientation

Select the primary orientation for the reinforcement.

Brkt Thickness

Select the thickness for the bracket.

Lapped

Select to create a lapped bracket system when a Sketched bracket symbol is selected. Lapped is automatically selected and unavailable if a lapped bracket symbol is selected.

Parent System

Select a parent system for the bracket plate system that you are placing. You can create new systems in the Systems and Specifications task.

Type

Select the type of plate system to place. If you do not specify a type, the software automatically determines the type based on the plate system orientation.

What do you want to do?

- *Add a flange to a catalog bracket system* (on page 1428)
- *Modify a bracket system* (on page 1429)
- *Delete a bracket system* (on page 1429)

Add a flange to a catalog bracket system

1. Click **Select bracket and flange symbols** .
2. In the **Bracket Symbol** box, be sure that the bracket catalog symbol chosen supports adding a flange.
3. In the **Flange Symbol** box, select **More...**.
4. Select a flange symbol to place from the catalog.

Modify a bracket system

1. Click **Select** on the vertical toolbar.
2. Select **Root Plates** in the locate filter.
3. Select the bracket to modify.

   TIPS

   - You can select the bracket in the model, or from the **Workspace Explorer**.
   - You can select and modify more than one bracket at a time. This is especially useful when you want to modify load points and offsets for multiple tripping brackets. For more information, see *Tripping Bracket Connect Points Data Tab (Bracket Plate System Properties Dialog Box)* (on page 1447).
   - Using the ribbon controls, edit the bracket as needed.

Delete a bracket system

1. Click **Select** on the vertical toolbar.
2. Select **Root Plates** in the locate filter.
3. Select the bracket system to delete.
4. Click **Delete** .

Bracket Plate System Properties Dialog Box

Specifies the properties for the bracket system that you are editing.

See Also

Bracket Flange Properties Tab (Bracket Plate System Properties Dialog Box) (on page 1446)
Bracket Parameters Tab (Bracket Plate System Properties Dialog Box) (on page 1445)
Configuration Tab (on page 124)
General Tab (on page 125)
Main Tab (Bracket Plate System Properties Dialog Box) (on page 1429)
Relationship Tab (on page 123)
Material Tab (Bracket Plate System Properties Dialog Box) (on page 1432)
Molded Conventions Tab (Bracket Plate System Properties Dialog Box) (on page 1432)
Tripping Bracket Connect Points Data Tab (Bracket Plate System Properties Dialog Box) (on page 1447)

Main Tab (Bracket Plate System Properties Dialog Box)

Specifies the general properties of the bracket plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

  - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
  - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** *Dialog Box* (on page 83).

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

Specifies the level of tightness as it applies to the entire plate system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and

reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* (on page 30) in the *Molded Forms User's Guide* or in the *Common User's Guide*.

See Also

Bracket Plate System Properties Dialog Box (on page 1429)

Material Tab (Bracket Plate System Properties Dialog Box)

Specifies the material properties for the plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Molded Conventions Tab (Bracket Plate System Properties Dialog Box)

Specifies the default plate and profile orientation rules for the bracket plate system.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.



Below - Positions thickness below the deck.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.



Down - Orients profiles below the deck. This is the default value.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.



Outboard - Orients profile flanges away from the port/starboard reference.



Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.



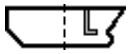
Starboard - Orients profile flanges to the starboard side.



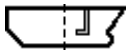
Transverse Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



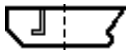
Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



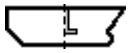
Aft - Orients profile flanges aft.



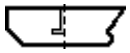
Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.



Outboard - Positions thickness away from the port/starboard reference.



Port - Positions thickness to the port side.



Starboard - Positions thickness to the starboard side.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.



Outboard - Orients profiles away from the port/starboard reference. This is the default value.



Port - Orients profiles to the port side.



Starboard - Orients profiles to the starboard side.



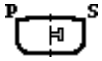
Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.



Down - Orients profiles downward. This is the default value.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



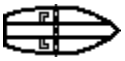
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.



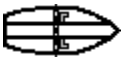
Aft - Orients profile flanges aft. This is the default value.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.



Aft - Orients profile flanges aft. This is the default value.



Transverse Bulkhead Molded Conventions

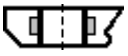
Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

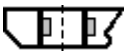
Inboard - Positions thickness toward the forward/aft reference.



Outboard - Positions thickness away from the forward/aft reference.



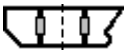
Forward - Positions thickness to the forward side.



Aft - Positions thickness to the aft side. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

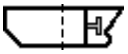
Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



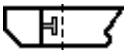
Aft - Orients profiles aft.



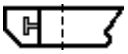
Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



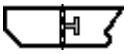
Aft - Orients profiles aft.



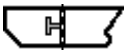
Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.



Aft - Orients profiles aft.



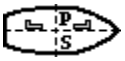
Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.



Starboard - Orients profile flanges to the starboard side.



Inboard - Orients profile flanges toward the port/starboard reference. This is the default value.



Outboard - Orients profile flanges toward the port/starboard reference



Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



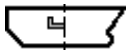
Starboard - Orients profile flanges to the starboard side.



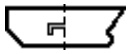
Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.



Down - Orients profiles downward.



Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.



Outboard - Positions thickness away from the port/starboard reference. This is the default value.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.



Outboard - Orients profiles away from the port/starboard reference.



Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.



Away From Centerline - Orients profile flanges upward and outboard.



Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.



Starboard - Orients profile flanges to the starboard side.



Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



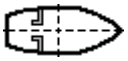
Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



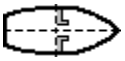
Aft - Orients profile flanges aft.



Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.



Aft - Orients profile flanges aft.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.



Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.



Centered - Centers thickness about the molded surface.



Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.



Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.



Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.



Right - Orients profile flanges to the right.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.



Bracket Parameters Tab (Bracket Plate System Properties Dialog Box)

Specifies the default values for the bracket plate system parameters. Parameter names and values are specific to each bracket symbol. Values can be modified as permitted by the catalog.

This tab is only visible after bracket creation.

Bracket Flange Properties Tab (Bracket Plate System Properties Dialog Box)

Specifies the default values for the bracket flange parameters. Values can be modified as permitted by the catalog. This tab is only visible after bracket creation and only if the bracket has a flange.

Molded Surface Side

Sets the molded side of the bracket and bracket flange. Supports creation of the flange symbol at the correct location while accounting for molded convention thickness direction and offset values.

- If the molded side is the inner surface, the value is 0.
- If the molded side is the outer surface, the value is 1.

Flange Width

Width of the flange measured from the edge of the bracket symbol, which is also the first bend tangent.

Plate Thickness

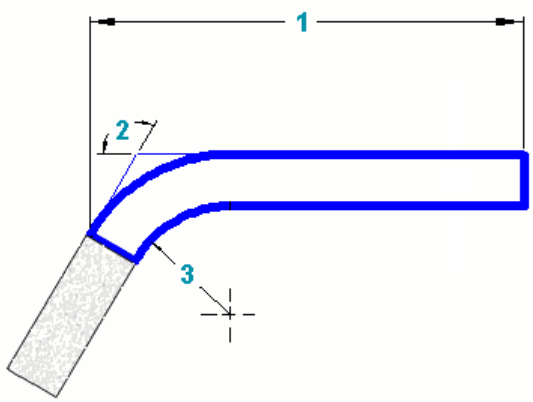
Specifies the thickness of the plate.

Bend Angle

Angle between the flange surface and bracket surface.

Inner Radius

Bend radius measured along the inner surface of the bend.



1 FlangeWidth

2 PlateThickness

3 BendAngle

4 InnerRadius

One option is also displayed.

Flange Orientation

Select the orientation of the flange bracket.

- **Same As Parent** - Orientation is the same as the bracket plate system primary profile orientation.
- **Against Parent** - Orientation is in the opposite direction of the bracket plate system primary profile orientation.

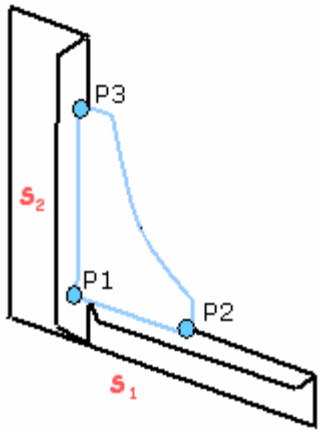
Tripping Bracket Connect Points Data Tab (Bracket Plate System Properties Dialog Box)

Specifies the default offset values and lapped status for a tripping bracket plate system. This tab is only visible after bracket creation.

Offset Values

Load Point

Specifies the support load point of each point used to calculate the bracket system plane. Offsets are measured from the load points. Load points are defined in the profile cross-section symbols. If one of the supports is a plate system, then the load point value is 0. For more information, see Key Points.



U Offset

Specifies the offset along the local U axis of the support profile. The value is 0 if the bracket is not on the flange of a profile support.

V Offset

Specifies the offset along the local V axis of the support profile. The value is 0 if the bracket is not on the flange of a profile support.

Override

Select to specify different values for **Load Point**, **U Offset** and **V Offset**.

Lapped Status

Displays the status of each tripping bracket support. **Lapped** is selected if the support is lapped. This value is set during bracket creation and cannot be changed here.

See Also

Bracket Plate System Properties Dialog Box (on page 1429)

Bracket Reinforcement Properties Dialog Box

Specifies the properties for the bracket system reinforcement that you are editing. Tabs on this dialog box are visible only after bracket creation.

Cross Section Tab (Bracket Reinforcement Properties Dialog Box) (on page 272)
Orientation Rule Tab (Bracket Reinforcement Properties Dialog Box) (on page 272)

Cross Section Tab (Bracket Reinforcement Properties Dialog Box)

Specifies the cross-section settings for a bucking stiffener bracket reinforcement. This tab is only visible after bracket creation when **Edge Reinforcement** or **Buckling Stiffener** is selected for **Reinforcement**.

Section Type

Specifies the profile cross-section type. The software displays the section type properties for the section type that you selected. Each section type has different properties.

Section Size

Specifies the profile section size.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Override Cross Section Information

Select to override the cross-section information generated by the rule.

WebLength(WL)

Displays the web length of the reinforcement.

WebThickness(WT)

Displays the web thickness of the reinforcement.

Orientation Rule Tab (Bracket Reinforcement Properties Dialog Box)

Specifies the orientation settings for a bracket edge reinforcement. This tab is only visible after bracket creation when **Edge Reinforcement** is selected for **Reinforcement**.

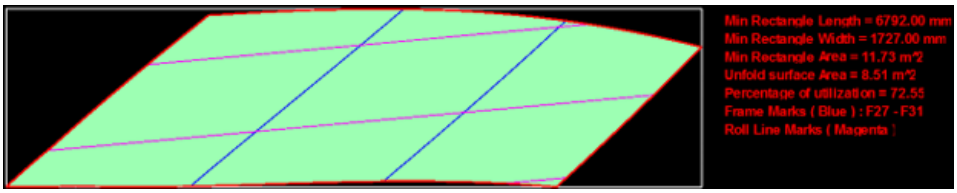
Rule

Select the orientation rule that you want to use for the edge reinforcement. Available rules include **Normal**, **Longitudinal**, **Vertical**, and **Transverse**.

APPENDIX D

Shell Plate Unfold Preview

Tools > Shell Plate Unfold Preview generates a graphic preview of an unfolded plate shape with frame marks, roll line marks, a surrounding minimum rectangle with annotated dimensions, an unfolded surface area, and the percentage of material used in a private graphic view.



Shell Plate Unfold Preview Ribbon

Select Plate System

Select the plate system that you want to preview.

Boundaries

Select the seams, frames, or split knuckles that intersect the selected plate system.

Finish

Click to display the unfolded output of the shell plate.

Solve Ambiguity

If the boundaries that you selected can form the object in more than one way, then you have defined an ambiguous solution. The software prompts you to select one or more solutions.

Reject

Click to rejects the plate selection.

Accept

Click to accepts the plate selection.

Frame System

Select the coordinate system to use for the unfolding.

1. Select **Tools > Shell Plate Unfold Preview**.
2. Select the plate to unfold.
3. Select the coordinate system from the **Frame System**.
4. Click **Finish**.

Glossary

abaft

Toward the stern of a ship, behind, further aft than.

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

access holes

An opening cut in the structure of a ship to permit entering or leaving various compartments.

accommodation

Functions for the design and construction of crew spaces aboard ship, particularly deck house design on commercial ships.

accommodation ladder

A portable set of steps on the side of a ship for the accommodation of people boarding from small boats or from a pier.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

aft

Toward, at, or near the stern.

after body

The hull from aft of the midship section.

aftermost

Nearest the stern.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

basic design

Engineering definition of the model and its systems.

bending

Process of conforming a plate to the hull surface. A bending process can be either simple (bent in one direction) or compound (bent in multiple directions). The software must minimize compound bending requirements.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

built ships

Complete database of NGC information after completion of the ship contract.

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

cabin plan

Deck plan arrangement of bulkheads, compartments, and furnishings.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

ceiling

Overhead design of the cabin area, including distribution systems for power, water, and ventilation.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

class

Grouping of individual objects that share some very significant, common characteristics.

class rule check

Verification that the developing design meets the rules of a particular classification society, such as ABS, Lloyd's, or DNV.

Class Rules

Classification Society Design Rules.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

control point

A point that is used to control the shape of a NURBS curve or surface. Curves have a one-dimensional array of control points, while surfaces have a two-dimensional array.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

corrugation

A surface or plate shaped into folds with a repeated pattern of parallel and alternating ridges and grooves, typically in an elongated V shape. In practice, a corrugated plate can be formed by swaging, but is more commonly bent through a series of alternating knuckles, such as those found on a corrugated bulkhead in an oil tanker.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

detail schedule

Lowest level of schedule used to manage and track work progress.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

edge reinforcement axis

A curve that you place along the edge reinforcement. This axis can exist as part of the free edge or an offset of a part of the free edge.

edge reinforcement boundary

An object that limits the edge reinforcement axis.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

external appendages

External structure attached to the hull, such as the propeller nozzle, shaft struts, bilge keel, and so forth.

fabricate

To cut, punch, and sub-assemble members in the shop.

face

A topological object that represents a trimmed surface bounded by a loop of edges.

face plate

An edge reinforcement type that places a plate or profile at the selected plate edge.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fair

To create a smooth line from one point to the next. In the case of a ship hull, to create a smooth line from one offset point to the next, or to create a smooth surface transition from one cross-section curve to the next.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

fire integrity

Deck and bulkhead treatments and fire and smoke blocks for fire control and retardation.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

focus of rotation

A point or line about which an object or view turns.

free edge

A plate edge that is not bounded by another structural object.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

girth

The transverse measurement around the surface of the hull. As used in the software, the measurement along the surface of a body in any direction.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

hull

The outside surface or envelope of a member.

hull applicability versioning

Capability within the database that allows the database to have a single model for a class of ships that are nearly identical (sister ships) while recognizing and managing the different parts in each ship of the class. The concept considers that alternate configurations are created for a state purpose (owner change order, new vendor, design improvement, and so forth). In addition, this concept relates the geometry and associated geometry to that purpose, and provides an extraction method from the database for viewing or manufacturing the base ship with the appropriate variant of the design authorized for that particular ship hull.

hull form generation development

Set of ship lines or surfaces that meet ship requirements for speed, powering, fuel rate, and cargo, and any special limitation of draft, beam, or length. At this stage, lines should be adequate for model tests.

hull structure design

Functions related to creating the structural design and manufacturing attributes of the hull and longitudinal and transverse structure of a ship.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

internal structure

All internal structural members of the ship.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

joiner

Non-structural bulkheads, and trim and built-in furnishings.

kinematics analysis

Analysis of mechanical motion.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual

Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance parts

Required material for depot or on-board repair or overhaul of equipment, as determined by engineering study. Generally at a level below the purchased construction object of the plant.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

natural surface

A surface without a boundary curve.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

non-aligned

If the tangents computed at intersecting points of plate to stiffen and supports are not parallel, then both the supports are called to be non-aligned.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

physical occurrence

Unique specific object that has traceability and is the physical manifestation of an occurrence object. A physical occurrence applies to one and only one hull. It is a version of its occurrence object with as-built or as-modified differences included and has a serial number or lot number.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

product

Data objects that describe the components of a ship and any corresponding properties. An individual object or part (or its representation in the product model) that may be installed in the ship. Examples of individual products include objects such as a coffee urn, a light fixture, a piece

of pipe, a piece of ventilation duct, a radar display console, a bulkhead plate, and a structural profile stiffening a bulkhead.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

query select sets

Set of objects that are selected in a query or queries on the database.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

sheetbody

A topological object that represents a collection of faces joined along their common edges (stitched).

shell structure

External portion of the surface of the plant.

ship

A collection of modeled objects that can be simultaneously displayed and edited in a workspace. A Ship points to a Catalog (optionally shared with other Ships). Access control is managed at the Ship level.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

specials

An option category that allows you to control specialized calculations for equipment trim, repeatability, and center-of-gravity.

specifications

Contracted requirements for the plant.

steel outfitting

Internal structural elements of a ship that are required to meet a local requirement such as foundations, non-structural bulkheads, walkways, and so forth.

stern frame

Casting and structure that support the rudder and shaft opening.

stud

A bolt, threaded on both ends, used to connect components.

surfaces

Those areas of a ship that include the hull, deck, and bulkhead surfaces.

suspended floor

A concrete floor system built above and off the ground.

swage

A surface or plate formed to a shape, typically a regular or non-regular pattern of grooves in an arc, V or elongated V shape. In practice, a swaged plate is formed by stamping or hammering with a tool, die, or press.

swash bulkhead

A longitudinal or transverse nontight bulkhead in a tank that decreases the swashing motion of the liquid contents. A plate in a tank that has this same effect but that does not extend to the bottom of the tank is called a swash plate.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

transverse

At right angles to the fore-and-aft center line.

transverse frames

The athwartship members that form the ribs of the ship.

trim

The difference between the forward draft and the aft draft.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

tumble home

The inboard slope of the side of a ship, usually above the designed waterline.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

vertical keel

A row of vertical plates extending along the center of the flat plate keel.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

water line

A line parallel with the base line that depicts the water.

watertight door

A door that when closed prevents the passage of water.

weather deck

A deck exposed to the weather.

weathertight door

A door that when closed prevents the passage of rain and spray.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

windlass

The machine used to hoist and lower anchors.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

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